AD-A078 376   CINCINNATI UNIV OH DEPT OF AEROSPACE ENGINEERING AND--ETC F/G 20/4
STUDY OF THREE-DIMENSIONAL INTERNAL FLOW PROBLEMS RELATED TO TU--ETC(U)
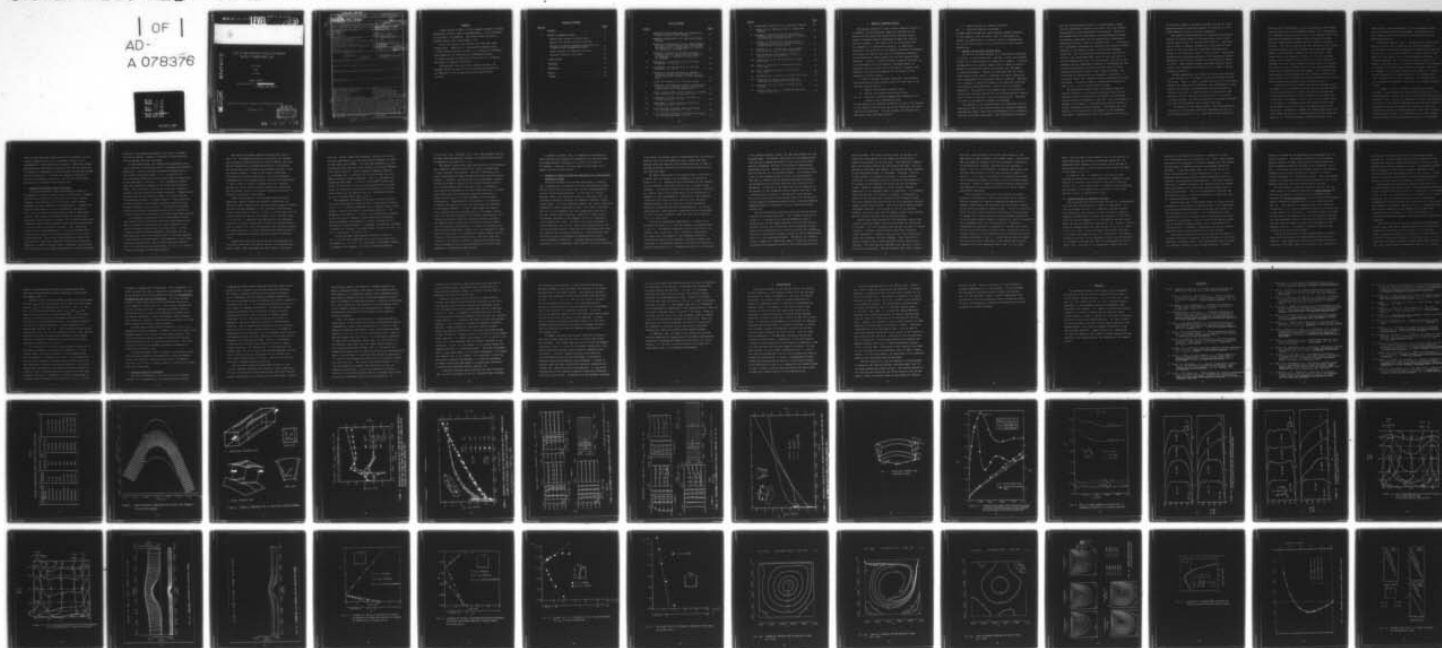FEB 79   K N GHIA , U GHIA   F49620-78-C-0041
UNCLASSIFIED   AFL-79-2-46   AFOSR-TR-79-1264   NL

1 OF 1
AD-
A 078376

END
DATE
FILMED
1 -80
DDC

REPORT NO. AFL 79-2-46

ADA078376

DDC FILE COPY

STUDY OF THREE-DIMENSIONAL INTERNAL FLOW PROBLEMS
RELATED TO TURBOMACHINERY FLOWS

K.N. GHIA

AND

U. GHIA

ANNUAL REPORT

AFOSR Contract No.
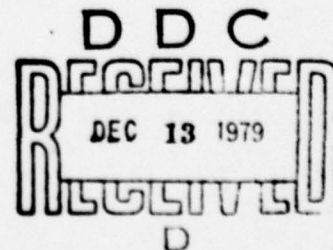
F49620-78-C-0041

February 1979

D D C
RECEIVED
DEC 13 1979
D

79 12 10 115

Unclassified

REPORT DOCUMENTATION PAGE	READ INSTRUCTIONS BEFORE COMPLETING FORM

1. REPORT NUMBER	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
AFOSR TR-79-1264		

4. TITLE (and Subtitle)	5. TYPE OF REPORT & PERIOD COVERED
STUDY OF THREE-DIMENSIONAL INTERNAL FLOW PROBLEMS RELATED TO TURBOMACHINERY FLOWS.	INTERIM Annual rept. 1 Jan 78 — 31 Dec 78
	6. PERFORMING ORG. REPORT NUMBER
	AFL-79-2-46

7. AUTHOR(s)	8. CONTRACT OR GRANT NUMBER(s)
KIRTI N. GHIA URMILA GHIA	F49620-78-C-0041

9. PERFORMING ORGANIZATION NAME AND ADDRESS	10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS
UNIVERSITY OF CINCINNATI AEROSPACE ENGINEERING & APPLIED MECHANICS DEPARTMENT CINCINNATI, OHIO 45221	2307A4 61102F

11. CONTROLLING OFFICE NAME AND ADDRESS	12. REPORT DATE
AIR FORCE OFFICE OF SCIENTIFIC RESEARCH/NA BLDG 410 BOLLING AIR FORCE BASE, D C 20332	1 Feb 79
	13. NUMBER OF PAGES
	66

14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office)	15. SECURITY CLASS. (of this report)
	UNCLASSIFIED
	15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public release; distribution unlimited.

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

THREE-DIMENSIONAL INTERNAL FLOW
LAMINAR, INCOMPRESSIBLE FLOW
POLAR CURVED DUCTS
VARIABLE-AERA DUCTS
NUMERICAL GENERATION OF COORDINATES

SPLINE DISCRETIZATION
VELOCITY-PRESSURE FORMULATION
ELLIPTIC, SEMI-ELLIPTIC AND
    PARABOLIZED NAVIER-STOKES EQUATIONS
FLOW SEPARATION, DIRECT POISSON SOLVERS

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

The primary objective of this two-year research project is to develop a finite-difference numerical technique for the solution of the parabolized Navier-Stokes equations using a system of surface-oriented coordinates suitable for ducts of arbitrary geometry. An additional objective to be fulfilled by this research consists of developing an analysis for laminar duct flow with streamwise separation using mathematical models simpler than the full Navier-Stokes equations. In order to achieve the above objectives, the following three problems have been studied during the first year of the two-year contract period: →next page

DD FORM 1473   EDITION OF 1 NOV 65 IS OBSOLETE   UNCLASSIFIED

410 677

ABSTRACT

A brief review of Task II of the research program on "Three-Dimensional Internal Flows in Turbomachinery," at the University of Cincinnati, is presented. The review covers the period between January 1978, and December, 1978.

During this period, five related areas have been under investigation. These are (1) Flow through Joukowski Ducts; (2) Flow through Curved Polar Ducts; (3) Analysis of Streamwise Flow Separation via a Semi-Elliptic Mathematical Model; (4) Higher Accuracy of Numerical Solutions and (5) Increasing Efficiency of Numerical Algorithms.

The personnel involved in this research program, the progress during the year as well as further work planned for the remainder of the contract period have been briefly outlined.

TABLE OF CONTENTS

LIST OF FIGURES

iv

1. REVIEW OF RESEARCH PROJECT

The primary objective of this two-year research project is
to develop a finite-difference numerical technique for the
solution of the parabolized Navier-Stokes equations using a
system of surface-oriented coordinates suitable for ducts of
arbitrary geometry. This objective is to be achieved within
the framework of a laminar-flow analysis. An additional
objective to be fulfilled by this research consists of developing
an analysis for laminar duct flow with streamwise separation
using mathematical models simpler than the full Navier-Stokes
equations. Finally, a finite-difference numerical technique is
to be developed using a two-equation turbulence model for
analyzing flows in straight and curved ducts of regular cross
sections, with particular emphasis on the accuracy of the
analysis in the wall region.

In order to achieve the above objectives, the following
three problems have been studied during the first year of the
two-year contract period:

    i. Laminar flow through Joukowski ducts,

    ii. Laminar flow through curved polar ducts,

and iii. Analysis of laminar streamwise flow separation via
         a semi-elliptic mathematical model.

In each phase of the research, attention is given to the accuracy
of the numerical solutions as well as the efficiency with which
they can be determined. For this purpose, the following two
additional areas have been studied:

1

iv. Higher accuracy of numerical solutions,

and v. Efficient numerical algorithms.

Suitable model problems have been selected, wherever required,
in this research, with the overall goal of contributing to
further the understanding of internal flows in turbomachinery.
Each of the above research areas is described in the following
subsections.

1.1 Laminar Flow Through Joukowski Ducts

The flow in the blade passages of a turbomachine (compressor
or turbine) is three-dimensional, unsteady, viscous and turbulent.
Further, the blade passages are geometrically complex. A thorough
understanding of this highly complex flow is needed in order to
achieve an optimum blade design and thereby gain improvement
in engine performance. Theoretical techniques that can fulfill
these objectives can serve to complement and partly replace the
costly experimental trial-and-error design procedures. Therefore,
it is necessary to develop theoretical and numerical techniques
for analyzing three-dimensional, viscous flow in non-simple
geometries. The purpose of the present study is to develop an
analysis and solution technique for the three-dimensional viscous
flow in ducts of arbitrarily varying cross sections. Therefore,
the study is applicable to cascade-like passages.

The geometry of the cascade channel is transformed from its
complicated shape in the physical domain to a simple rectangular
computational domain where the flow boundaries align conveniently
with the transformed coordinates. The coordinates are adjusted

2

such that one coordinate direction is approximately aligned
with the predominant direction of the flow; this later permits
parabolization of the transformed flow equations. Because of
the complex nature of the geometry, the transformation between
the physical coordinates and the computational coordinates is
established through a numerical procedure (Ref. 1). Accordingly,
the transformed coordinates are obtained as solutions of Poisson
equations in the physical domain, with Dirichlet boundary
conditions. Even the coordinate-transformation equations are
solved in the transformed domain so as to allow the use of a
uniform computational grid. For this purpose, the role of the
dependent and the independent variables is interchanged and the
inverted-transformation equations are then solved using an
alternating-direction implicit (ADI) numerical method. The
coordinates so generated are not necessarily orthogonal. In
the present study, only radially invariant geometries have been
considered. A possible system of transformed coordinates is
shown in Fig. 1 for a typical turbine cascade.

The mathematical model of the flow problem is derived
from the Cartesian- or cylindrical-coordinate continuity and
momentum equations transformed to the general non-orthogonal
surface-oriented coordinates for the duct. The flow problem is
assumed to be parabolic in the main-flow direction because of
minimal streamwise viscous diffusion. This permits the flow
equations to be parabolized with respect to the streamwise
coordinate. These equations then lend themselves to solution

by a marching numerical procedure, thereby avoiding the large
computer-storage and time requirements for solution of a full
three-dimensional elliptic system of equations.

The numerical method employed for obtaining the solutions
consists of the ADI scheme in the cross-plane, with a Crank-
Nicolson type marching in the main-flow direction. Particular
care is taken to preserve conservation of the mass-flow rate
across the duct even when the transformed cross-sectional
surface is not completely normal to the duct axis. The Neumann
boundary-value problems encountered in the analysis are solved
numerically using the SOR (Successive Over-Relaxation) method
by points. These Neumann problems have been carefully formu-
lated in the transformed coordinates so as to obtain a rapidly
converging solution.

Neither experimental nor theoretical results are available
in the literature for 3-D laminar viscous flow through varying-
area ducts. Hence, in order to assess the validity of the present
analysis and computational procedures, the incompressible viscous
laminar flow through a two-dimensional diverging channel is
used as a test case. An exact solution of the Navier-Stokes
equations for this problem is available (Ref. 2) in a polar
coordinate system. The results obtained by the present approach
show very good agreement with the exact solutions.

The analysis is then used to predict the three-dimensional
viscous flow in Joukowski ducts with rectangular as well as
polar cross sections (Fig. 2). This term has been adopted to

4

define a duct of rectangular or polar cross-section whose aspect ratio and area vary with streamwise distance due to the presence of a Joukowski airfoil profile on two of its opposite walls. The results are presented in terms of axial variation of primary velocity, mean-viscous-pressure drop and cross-flow velocity profiles. The results obtained are also analyzed for the various problem parameters such as the aspect ratio, Reynolds number based on inlet hydraulic diameter and the area ratio. Some sample results are presented in Figs. 3-5. In all the cases studied, the constant-area duct beyond the airfoil section is extended until the fully developed velocity profile is obtained. The case of a polar Joukowski duct is a very good approximation of an actual turbine-stator blade passage. For a typical polar configuration, the development of axial velocity profiles, the axial variation of the centerline and the maximum axial velocities and the mean-viscous pressure drop are shown in Fig. 6.

The results for both rectangular and polar Joukowski ducts are found to be in conformity with the general physical features of the flow and serve as a strong indication of the adequacy of the present analysis of flow in geometrically complex con-figurations. For example, in the vicinity of the leading edge of the airfoil, the presence of strong secondary flow causes the streamwise velocity to attain its maximum value off the centerline. Consequently, between the leading-edge section and the minimum-area section, the axial-velocity profile shows two peaks in it. Also, as expected, the Reynolds number is no

5

longer simply a scaling parameter for the streamwise distance even for the axial-velocity development. Nevertheless, the axial velocity as well as the mean viscous-pressure drop are weak functions of Re.

A technical abstract (Ref. 3) summarizing the computational details of this study is submitted for possible presentation at the Second International Conference on Mathematical Modelling in July 1979, in St. Louis, Missouri; a written version of the paper will then appear later in the Conference Proceedings. A technical abstract (Ref. 4) containing the main body of the results of the physical flow problem is submitted for presentation at the ASME 1979 Winter Annual Meeting to be held in New York in December 1979; a written version of that paper is to appear in the Proceedings of an ASME Symposium. Yet another paper (Ref. 5) based on this work is presently in preparation and is to be submitted for possible publication in a refereed technical journal.

In summary, it may be therefore stated that the study provides a viable approach for the accurate prediction of the three-dimensional viscous flow field, including the recirculating secondary flows, in ducts with arbitrarily varying cross-sections. Although only the laminar incompressible case has been presently considered, the analysis serves as a significant basic step towards the analysis of three-dimensional viscous flow fields in turbomachinery cascades. For applicability to a real cascade, several extensions and generalizations need to be considered.

These include additional complications in the geometry such as
turning, stagger and twist for the blades, as well as the
implementation of the periodicity condition. Additional effects
associated with the physical features of the flow are those due
to turbulence, compressibility and upstream influence and stream-
wise flow separation. All these are important effects which
can now be considered, one by one, with the help of the basic
building block developed in the present work.

1.2 Laminar Flow Through Curved Polar Ducts

A large and important class of engineering problems is
represented by internal viscous flows in curved ducts. Accurate
and reliable calculation of such flows is of great practical
interest in the prediction of flow through inlets, nozzles,
diffusers and other components of turbomachinery. Curved ducts
of polar cross section (i.e., finite-annular cross section;
Fig. 7) are of particular significance in many of these appli-
cations. When viscous fluid flows through the curved ducts,
centrifugal forces induce a secondary flow in the cross-sectional
planes and lead to higher pressure losses than those encountered
in ducts without longitudinal curvature. Thus, curved-duct
flows are truly three-dimensional, so that their determination
requires the solution of the three-dimensional elliptic Navier-
Stokes equations. While analytical solutions for these flows
are impossible, even their numerical solutions can become very
expensive in terms of computer costs. Therefore, other methods
are sought which involve some simplification of the Navier-Stokes

7

equations, with minimum compromise on the class of problems that can be treated. Numerical solutions of these equations can be obtained with great efficiency.

In the present study, the complex three-dimensional flow field occurring in curved polar ducts is analyzed and determined numerically. The flow problem is modelled mathematically in the toroidal coordinate system (r, θ, ϕ), as shown in Fig. 7, using the three-dimensional Navier-Stokes equations parabolized in the longitudinal direction ϕ. Thus, streamwise diffusion terms are neglected as compared to the cross-plane diffusion terms. This also leads to decoupling of the axial pressure gradient from the cross-plane pressure gradients. Parabolization requires that the flow Reynolds number is not small and that the streamwise velocity is nowhere negative. These conditions are well satisfied in the present problem provided the longitudinal curvature is not too sharp. This formulation permits a marching calculation procedure starting from a given set of initial conditions to the solution at desired downstream locations.

For adequate representation of the high-gradient regions near the duct walls, analytical coordinate transformations are used to introduce a nonuniform grid, thus circumventing the need for excessive computational time and storage. A transformation, which had been successfully used earlier for the two-dimensional cavity-flow problem (Ref. 6), has been employed in the present duct-flow problem. The parameters in this transformation provide control over the first step near the wall. Thus, the first step can be varied from $1/(2N)$ to $1/(8N)$, where N is the total number of intervals employed.

8

The solution procedure employed parallels that used in Refs. 7-9. The momentum equations are solved by the Douglas-Gunn two-step alternating-direction implicit method, while the Neumann boundary-value problems encountered in the analysis are solved using the successive over-relaxation method. Since the Poisson operator in terms of the present transformed coordinates has varying coefficients, the finite-difference representation of the Neumann problem required an extreme amount of care in order to obtain a converged solution. Accurate satisfaction of an integral relation between the source term and the boundary value of this Neumann problem was crucial to obtaining a solution of the problem.

First, the reliability of the present analysis and the associated computer program was checked by reproducing the results for some cases for which solutions were already available. This also allowed assessment of the coordinate transformation. Figure 8 shows typical results for flow in a curved polar duct and a curved rectangular duct of unity aspect ratio, for Dean number $K = 55$ and 100. A comparison has been made for the streamwise variation of the axial centerline velocity and the mean-pressure drop for these ducts. Both configurations show identical behavior for these quantities. Similar conclusions were observed earlier for the corresponding straight polar and straight rectangular ducts.

Results have been obtained for several configurations of the curved polar duct, with various values of the problem parameters, namely, Reynolds number Re, Dean number K, and aspect

9

ratio AR. Figure 9 shows the streamwise variation of the skin-friction coefficient C_{f_w} at the two positions denoted by A and B for the cross section shown. For each case presented, C_{f_w} has its maximum value near the initial station with uniform entrance flow velocity profile. With increasing streamwise distance, C_{f_w} decreases until it achieves its fully-developed value and remains constant further downstream. Also, C_{f_w} decreases with increasing Dean number. In view of the nature of centrifugal forces, the position of B on the outer surface with the larger radius of longitudinal curvature and the position of A on the inner surface with the smaller radius of transverse curvature, C_{f_w} at B is always larger than C_{f_w} at A.

The streamwise development of the axial velocity profiles along the mid-surfaces of the polar-duct cross section is shown in Fig. 10. It should be observed that no symmetry exists across either of these mid-surfaces in the curved polar duct. The asymmetry with respect to θ is due to the longitudinal curvature of the duct and is, therefore, significantly more pronounced than the asymmetry with respect to r which is mainly due to transverse curvature effects. As expected, the degree of asymmetry increases with increase in the Dean number K as seen from Fig. 10a for K = 100 and Fig. 10b for K = 630.

Typical secondary-flow velocity profiles are shown in Fig. 11 for the fully developed flow in two curved polar duct configurations. At K = 100, Fig. 11a shows a considerable degree of asymmetry with respect to θ; this is further enhanced at

10

K = 630 (Fig. 11b). Moreover, at K = 630, the secondary velocity profiles indicate additional changes in secondary-flow direction as compared to the case for K = 100.

The results indicate that the study provides a valid analysis and a numerical method of solution for flows in curved ducts of great practical significance. The use of analytical coordinate transformations makes the method efficient and the results accurate. Presently, the additional pair of secondary vortices indicative of Dean's instability for curved rectangular ducts have not been identified for the polar case. A systematic study should be performed for the fully developed flow in curved polar ducts using the appropriate asymptotic equations to analyze the possibility of this phenomenon for the present configuration. These asymptotic solutions will also serve as a check on the accuracy of the fully developed flow generated by the present analysis. In fact, these asymptotic results would comprise the only available check on the present results since, to the best knowledge of the present investigators, flow through curved polar ducts has not been previously analyzed. The present study provides the first set of available results for this important configuration. To render this research even more physically meaningful, results are to be generated for cases where the outer wall $r = R_2$ rotates at constant angular velocity. Finally, a rotation of the entire duct, with this outer wall rotating in the opposite direction, should also be considered. This configuration very closely approaches the rotating blade passages in turbomachinery applications.

A technical abstract (Ref. 10) summarizing the results of this study has been accepted for presentation at the AIAA Fluid and Plasma Dynamics Conference to be held in July 1979 in Williamsburg, Virginia; a written version of the paper is to appear later as an AIAA Preprint.

1.3 Analysis of Laminar Streamwise Separation via a Semi-Elliptic Mathematical Model

Sufficient evidence has now been established to indicate that incompressible flow separation can be represented by mathematical models that are simpler than the full three-dimensional elliptic Navier-Stokes equations. In the limit of infinite Reynolds number, the triple-deck theory of Stewartson (Ref. 11) provides an asymptotic analysis for separation bubbles of small normal extent. For finite Reynolds numbers, the various simplified mathematical models considered in Refs. 12-14 lead to solutions that are in good agreement with the corresponding solutions of the full Navier-Stokes equations. However, all these finite-Re analyses have been in the framework of the ω-ψ formulation for two-dimensional flows. In view of the growing interest in three-dimensional flow and the difficulties associated with the vorticity-stream function formulation for three-dimensional flows, the present study is aimed at developing a simplified model and an efficient solution procedure for separated flow using a velocity-pressure formulation. Although results are presently obtained for a two-dimensional model problem, the entire philosophy

12

in the study is directed toward a three-dimensional flow analysis. Moreover, for this two-dimensional case, results were also obtained using the complete Navier-Stokes equations in order to provide an accurate evaluation of the solutions resulting from the simplified model.

The model problem used in the present study is the flow in a doubly infinite channel with an asymmetric distortion as shown in Fig. 12. Surface-oriented coordinates for this channel are the conformal coordinates obtained via a Joukowski transformation. The straight channel with parallel walls is obtained as a special case of this configuration by the variation of a parameter in the transformation. Clustering of the coordinate curves near the channel walls as well as in the regions of maximum curvature near the lower wall is also achieved by analytical coordinate-stretching transformations. For certain configurations, three regions of separated flow are anticipated to occur for such a geometry; these are indicated as regions 1, 2 and 3 in the figure.

The flow is represented mathematically in terms of the primitive variables (v,w,p) and the conformal coordinates shown in Fig. 12. The momentum equations governing the velocity components v and w along the η and ζ directions, respectively, are parabolized with respect to the streamwise direction ζ. The governing equation for pressure is derived by forming the divergence of the momentum equations, and using the continuity equation indirectly in it. Streamwise ellipticity is maintained

in the pressure equation; hence, the term semi-elliptic for the proposed model. The boundary conditions for the momentum equations are quite simple and consist of zero slip at the walls and zero streamwise variation of velocities at upstream and downstream infinity. However, any realistic velocity profile may be prescribed at the upstream boundary. The boundary conditions for the pressure equation are not so straightforward. They must be derived from the momentum equations evaluated at the boundaries while strictly enforcing zero dilation at all boundaries. Evaluation of the normal derivatives appearing in the equation at the boundaries requires special care as it has a non-negligible influence on the solution in the interior. The initial conditions for the problem consist of the solution of the parabolized momentum equations with a uniform streamwise-pressure gradient corresponding to fully developed straight-channel flow.

The finite-difference equations resulting from the discretization of the quasilinearized η- and ζ-momentum equations on curves of ζ = constant are solved simultaneously using a generalization of the Thomas algorithm; this has been found to be more efficient than conventional block-tridiagonal solvers. Starting with prescribed inlet velocity profiles, the streamwise marching of the solution proceeds through the entire computational field up to the downstream boundary. Thereafter, the pressure field is updated by a prescribed number of iterations using the Successive-Over Relaxation method or the Alternating-Direction

14

Implicit method. The local dilation as well as the mass flow across each cross section of the channel are monitored as a check on the accuracy of the solutions. Some additional remarks about the pressure solution are worth mentioning here. For flow through an infinitely long channel, the pressure becomes unbounded as the downstream boundary is approached. Moreover, since the Poisson equation for pressure is subject to Neumann boundary conditions, its successful solution requires the satisfaction of an integral relation between the source term and the boundary values. Numerical consideration of this relation becomes impractical for an unbounded domain of integration. To circumvent these difficulties, the problem for pressure is formulated in terms of a new variable that represents the difference between the actual pressure distribution and the fully-developed straight-channel pressure distribution. This new variable remains bounded everywhere; moreover, it can be set to zero at the upstream boundary, thus doing away with the Neumann boundary value problem and the integral relation over an unbounded domain.

Results were first obtained for the degenerate case of flow through a straight channel with fully developed flow conditions at entrance. The final converged solution was obtained in a single iteration. Moreover, forcing several additional iterations showed imperceptible changes in the solution, thus indicating a stable and convergent solution procedure. Next, the flow in a straight channel with a uniform entrance velocity profile was computed. This case was also computed using the fully elliptic Navier-Stokes equations. The two solutions were found to agree

15

very well, thus confirming the validity and accuracy of the semi-elliptic model proposed in the present study. Calculations were then made for channels with various degrees of distortion. As the distortion increased, reversed flow was observed in region 1 (Fig. 12) of the channel. Fine-mesh calculations also indicate the existence of a very small separation bubble in region 2. In all cases, the results of the semi-elliptic model are in excellent agreement with those of the full Navier-Stokes solutions. Figure 13 shows the streamline pattern obtained from the computed velocity-pressure solution for a typical channel encountering separation.

Thus, the study is believed to have served its primary objective of formulating a semi-elliptic model and an accurate solution procedure for laminar incompressible separated flow. However, these results are considered only preliminary; several points are being presently studied further. So far, no particular effort has been directed toward the efficiency of the solutions. It appears that a more consistent finite-difference formulation of the velocity-pressure system is possible with use of a staggered grid arrangement than with the regular grid. Therefore, a staggered-grid formulation is expected to increase the accuracy as well as the efficiency of the solution. Significant gain in solution accuracy is also expected by using the simplified spline procedure described in Ref. 15. Higher-order accurate fine-mesh solutions will enable comparison of the present work with the asymptotic solutions obtained by Smith (Ref. 16) for channel

flows. The true test of this research lies in the analysis of three-dimensional duct flow with streamwise separation. An appropriate model problem for this purpose can be obtained by placing bounding walls a finite distance apart across the width of the channel in Fig. 12.

An abstract (Ref. 17) summarizing this work has been accepted for presentation at the Symposium on Computers in Aero-dynamics to be held at the Polytechnic Institute of New York in June 1979. A full length paper is presently under preparation and is planned to be submitted for publication in the International Journal on Computers and Fluids.

1.4 Higher Accuracy of Numerical Solutions

The two-dimensional Navier-Stokes equations for incompressible flow have been investigated by a number of researchers. The vorticity-stream function (ω, ψ) formulation has been studied very extensively using explicit as well as implicit numerical schemes of second-order or fourth-order accuracy. On the other hand, the velocity-pressure (u, v, p) formulation had been solved mainly by explicit numerical procedures so far. Recently, implicit solution procedures have also been applied to the (u, v, p) formulation (e.g., Ref. 6) with efficiency and accuracy comparable to or better than those for the (ω, ψ) formulation. The primary difficulty with the (u, v, p) formulation lies in the determination of a pressure field that can lead to a divergence-free velocity field. But, it is the (u, v, p) formulation, rather than the (ω, ψ)

formulation, that admits more readily to extension to flows with three-dimensionality, turbulence, etc. With a trend towards analysis of flow problems of increasing complexities, the accuracy and efficiency of numerical solutions have become exceedingly important. It has been shown (Refs. 18, 19) that the use of spline approximations in the discretization of the (ω, ψ) differential equations has several important advantages over conventional finite-difference approximations. Derivative boundary conditions, irregular boundaries and nonuniform mesh are handled better and extension to higher-order accurate formulations is more straightforward. Even for the second-order accurate form, the truncation error of a spline formulation is lower than that for other second-order schemes.

The principal goal of the present study is to employ spline polynomials in the numerical solution of the (u, v, p) formulation of the Navier-Stokes equations and to analyze the accuracy and the efficiency of the solutions. The incompressible laminar viscous driven flow in a square cavity is selected as the model problem for this purpose. This is a true Navier-Stokes problem with no admissible boundary-layer-like or other simplifying characteristics. Furthermore, experimental as well as accurate numerical results are available for this flow problem.

The governing equations for this problem are the momentum and mass conservation equations; only the convective form of the equations is presently used. Since the pressure does not appear as a dominant variable in any of these equations, a Poisson

equation is formed for the pressure using the momentum and continuity equations. The boundary conditions for the pressure equation consist of normal derivatives of pressure determined from the momentum equation evaluated at the boundaries. As in Ref. 6, the source term and the boundary values in this Neumann boundary-value problem are required to satisfy an integral relation necessary for a convergent solution. Also, the pressure equation is formulated directly from the discretized form of the momentum equations using the various spline relations.

For the purpose of efficient numerical solution, the non-linear terms in the momentum equations are quasilinearized and the resulting equations together with the necessary spline relations are solved simultaneously using an alternating-direction implicit scheme and a generalization of the Thomas algorithm (Ref. 20). This approach is computationally much faster than conventional (4 x 4) block-tridiagonal solvers and is stable over a wider range of the time step Δt. Even when optimal time steps are employed in the momentum as well as the pressure equations, the time-independent nature of the continuity equation suggests that more than one pressure iteration be performed for each momentum iteration. The number of pressure iterations used varies with the grid size employed.

The spline formulation of the pressure equation presents some unique difficulties not experienced with the vorticity equation. Even though both ω and p and their derivatives are singular at the upper corners of the cavity, the ω-equation is

19

not required at these singular points, whereas the pressure
equation is. Various formulations have been attempted for the
pressure derivatives at these singular points, including a cubic
extrapolation from the neighboring points. Considerable improve-
ment is possible in this area by asymptotic analysis of the
corner singularities as explained later in this section. Never-
theless the approach used presently yields reasonably accurate
results, with the velocity divergence in the order of 10^{-4}.

Results have been obtained for several values of Re up to
1000 and compared with other available finite-difference solutions
as well as spline solutions of the (ω,ψ) formulation. As indicated
in Ref. 19 for (ω,ψ) spline solutions, the present (u,v,p) spline
solutions, with the grid sized used, are more accurate than
corresponding finite-difference solutions even with a finer grid.
For purpose of an accurate comparison, the (ω,ψ) spline solutions
were actually computed, by the present authors rather than
obtained directly from Ref. 18. Having the complete (ω,ψ)
solution available also enables a more detailed and comprehensive
comparison.

Since the behavior of the flow variables for this problem
is well known, the results presented here are only those corre-
sponding to the analysis of the accuracy of the solutions.
Figure 14 shows the variation of vorticity ω_c at the midpoint
of the moving wall with variation in mesh size Δx $(= \Delta y)$. All
three sets of results presented indicate second-order accuracy.
The (u,v,p) spline solution agrees well with the (ω,ψ) spline

20

solution, and both these solutions have much smaller error than the finite-difference solution obtained by the factored ADI scheme of Ref. 21.

Figures 15 and 16 show the effect of mesh size on the stream function ψ_{max} at the vortex center of the cavity and the minimum velocity u_{min} at the centerline of the cavity. The (ω, ψ) and the (u, v, p) spline formulations lead to ψ_{max} and u_{min} that agree only at the limit of vanishing step size. In one formulation, the limiting value is approached from above while, in the other, it is approached from below. The pressure computed in the (u, v, p) spline formulation is also indicated to be second-order accurate (Fig. 17). Also included here are typical contour plots (Fig. 18) for the cavity flow at Re = 1000, showing that the (u, v, p) spline formulation developed in the present research is capable of computing this high Re case correctly even with a uniform (33 x 33) grid.

It is important to point out that although the behavior of the solution appears to be linear with respect to Δ^2 in the figures shown, an actual calculation of this exponent shows that the exponent is frequently far from 2. This observation has also been confirmed for several other available solutions that graphically indicated second-order accuracy. The authors believe that the departure from exact second-order accuracy is primarily due to the singularities that exist at the two upper corners of the cavity. Therefore, an effort has been made to consider nonuniform motion of the upper wall (e.g., $U = \sin^2 \pi x$)

in order to eliminate the singularities. More importantly, an
effort has also been made to reformulate the dependent variables
of the original cavity-flow problem such that the new dependent
variables are free from the singularity. The local singular
solution is the Stokes-flow solution determined analytically
as an asymptotic series (Ref. 22). This solution vanishes far
from the corner points and is therefore subtracted from the
original solution to yield the new dependent variables which
are regular everywhere. Results are presently being computed
using these new variables. These results are expected to be the
most accurate and efficient results possible.

A fourth-order accurate spline formulation of the (u,v,p)
system has also been attempted. Presently, the source term in the
pressure equation is not exactly fourth-order accurate. The
difficulty arises primarily because the formulation requires
pressure derivatives at the singular corner points. The intro-
duction of non-singular dependent variables is expected to
relieve this difficulty also.

A technical abstract (Ref. 23) based on the results of this
study has been accepted for presentation at the AIAA Computa-
tional Fluid Dynamics Conference to be held in Williamsburg,
Virginia, in July 1979; a written version of the work will appear
later as an AIAA paper.

1.5 Efficient Numerical Algorithms

The velocity-pressure (u,v,p) and the vorticity-stream
function (ω,ψ) formulations of the two-dimensional unsteady

22

incompressible Navier-Stokes equations have been solved by com-
bining two very efficient and accurate numerical techniques.
The velocity transport equations in the primitive-variable
(u,v,p) formulation and the vorticity transport equation in
the vorticity-stream function formulation are solved by employing
a modified form (Ref. 6) of the alternating-direction implicit
(ADI) method of Peaceman and Rachford. In addition to these
transport equations, both formulations contain an elliptic
Poisson equation with either Dirichlet or Neumann boundary
conditions. By far, the greatest computational effort is
expended in obtaining numerical solutions of these Poisson
equations. This is because these equations are truly elliptic
and their solution time increases rapidly with the number of
computational points employed. For 3-D unsteady viscous flow at
high Reynolds number, reliable numerical solutions must employ a
reasonably fine computational mesh. Hence, it is necessary
to develop very efficient algorithms for solution of these
Poisson equations. The Hockney (Ref. 24) and especially the
Buneman (Refs. 25, 26) Direct Poisson solvers represent two such
highly efficient solution techniques for the standard "five
point star" discretization of Poisson   equations with either
Dirichlet or Neumann boundary conditions.

The nonlinear convection derivatives occurring in the
velocity transport equations are treated by quasi-linearization
in time, thus retaining formal second-order space-time accuracy
without the necessity of iterating the nonlinear terms. However,

23

the nonlinear terms in the vorticity transport equation are merely linearized through a time delay of the stream function, thus resulting in a difference equation which possesses second-order spatial accuracy but only first-order temporal accuracy. Here, the temporal second-order accuracy is achieved by iterations. Also, both conservative and convective finite-difference forms of the transport equations are investigated with regards to computational efficiency and the accuracy of the numerical solutions obtained.

Thus, the present study consists of a detailed numerical investigation of the solution of a model two-dimensional laminar incompressible viscous flow configuration by a total of eight numerical procedures (see Table 1) using finite-difference mesh spacings of (17 x 17) to (65 x 65) grid points, over a range of Reynolds numbers from 100 to 1000. For a meaningful comparison of these various procedures, Roache (Ref. 27) has pointed out the essential requirement that the common testing ground be a truly Navier-Stokes model problem, that is, one without any admissible boundary-layer or other such simplifying characteristics. Hence, it was decided to investigate the problem of shear-driven flow within a two-dimensional square cavity as the model problem for this study also. The corner singularities present in this flow configuration do present some theoretical as well as numerical difficulties, in that, the numerically estimated accuracy of the results reported herein is degraded below second order as one approaches either of the singularities. To

circumvent this difficulty, some effort has again been made to obtain solutions using a modified velocity distribution at the moving-wall boundary, namely, $u = \sin^2 \pi x$, as this case is expected to be free of singularities.

Numerical solutions for the vorticity, stream function, velocity, and pressure fields have been obtained for the cavity flow starting from a no-flow condition and proceeding to the fully developed recirculating steady-state flow as illustrated in Fig. 19. Here, τ represents a nondimensional unit of time. To facilitate the comparison between solutions using the vorticity-stream function (ω, ψ) variables and those utilizing the velocity-pressure (u, v, p) variables, a regular grid system is presently employed. Typical results for the axial velocity profile through the vertical centerline of the cavity have been plotted in Figs. 20 and 21. As seen in Fig. 20, the results of the present study using the (ω, ψ) system, with uniform (33×33) and (65×65) grids, are in good agreement with the results available in the literature. In Fig. 21, the present (u, v, p)-system results are compared with the (ω, ψ)-system solutions. The slight discrepancy seen in this figure seems to be a direct consequence of the coarse grid used for the (u, v, p) system. It is important to note that the current convective ADI-Buneman (ω, ψ) solution represents an order of magnitude improvement in computational efficiency over previously published results (see Ref. 28).

A heuristic accuracy study following the lead of Blottner (Ref. 29) and illustrated in Fig. 22, indicates that the formal

25

second-order spatial accuracy of these Navier-Stokes solvers degrades somewhat as one approaches the locations of the corner singularities of this model problem. This heuristic accuracy analysis uses Richardson's extrapolation method and the results for the (17 x 17), (33 x 33), and (65 x 65) grids to estimate the infinitesimal-grid solution. This infinitesimal-grid estimate is then employed to yield an estimate of the percentage error level existing in a particular solution. A log-log plot of the absolute values of the percentage error versus number of grid points along the x-axis is then executed. If these points lie along a straight line whose slope is -2, then the assumed second-order spatial accuracy is confirmed; deviation from this straight-line behavior represents degradation in the accuracy level.

In summary, efficient and accurate numerical solutions of the Navier-Stokes equations for flow in a driven cavity have been obtained. The Poisson equations with Dirichlet and Neumann conditions in the (ω,ψ) and (u,v,p) systems, respectively, have been solved using the fast direct methods of Buneman as well as Hockney. A careful comparative study of the numerical solutions obtained with various algorithms is carried out; extensive results are presently being generated. A couple of problem areas have been identified as potential candidates for further work; these are briefly outlined here: (i) The corner singularities can be removed by employing analytical asymptotic solutions at the corner. Introduction of new dependent variables

26

will not only improve the accuracy but will aid also in improving the efficiency of the solution. Some effort has already been initiated towards this analysis. (ii) The (u,v,p)-system solutions are sought with a view that this system is easily extendable to three-dimensional flows. However, the present solutions show that the dilation $D \triangleq u_x + v_y$ does not approach a desired low level. It now appears that use of a "staggered" grid as suggested by researchers at Los Alamos Laboratory (Ref. 30) will lead to desired level of dilation in the field. Based on the study carried out so far, three different types of formulations for the discretized problems are identified. These are associated with achieving desired low levels of dilation and are referred to as weakly, mildly and strongly solenoidal formulations; these are currently being studied further.

An abstract (Ref. 31) based on the results of this work has been submitted for presentation at the Second International Conference on Mathematical Modelling to be held in St. Louis, Missouri, in July 1979; a written version of the paper will then appear later in the Conference Proceedings.

2. FUTURE EFFORT

During the remainder of the contract period, the various projects described in the preceding section will be continued and brought to a state of completion. This will include the preparation of technical reports or papers describing the research in reasonable detail, in addition to completing the research itself. For example, in the study of duct flows, rotation of the outer wall for curved polar ducts will be considered in order to simulate a rotating annular cascade and study the effect of this wall rotation on the flow in polar ducts. In the area of improvement of the accuracy of numerical solutions and the efficiency of solution algorithms, an effort will be made to develop an asymptotic solution for the flow near the upper corners of the driven cavity. This will lead to the formulation of new dependent variables that are non-singular at these corners. Computation of the flow solution in terms of these singularity-free variables is expected to lead to a more accurate and efficient numerical solution. For the same purpose, the cavity-flow configuration with $U = \sin^2 \pi x$ will also be analyzed more fully as this configuration does not contain any singularities. Finally, a staggered-grid finite-difference discretization as well as a fourth-order accurate spline formulation of the (u,v,p) system will be studied further in order to obtain accurate flow solutions of the cavity-flow problem with minimum dilation for the velocity field.

28

The work discussed so far is for laminar flow. However, most flows in practical applications are turbulent in character. Therefore, a concurrent effort has been also initiated for the study of turbulent flow. This effort is presently at a rather preliminary stage; hence, it was not included in the discussion in the preceding section. The two-equation turbulence model of Launder and Spalding (Ref. 32) has been used by Sokhey, K. Ghia and U. Ghia (Ref. 33) to study the three-dimensional turbulent flow inside curved ducts of regular cross-sections, with the proper wall functions for the regions near the duct walls. However, the accuracy as well as the efficiency of these numerical solutions needs to be studied in detail. Analytical transformations of the form used in Ref. 6 will be used to provide the desired clustering of grid points in the regions of high gradients near the walls. Solution of the flow equations and, in particular, of the Poisson equation for pressure with Neumann boundary conditions, in these transformed variables is not a trivial task. The results of the turbulent flow cases will be verified with the experimental data in order to demonstrate, in a convincing manner, the usefulness of the turbulence model and the associated analysis developed.

The wall regions will also be treated in a unified manner by employing the Jones and Launder turbulence model (Ref. 34) for these low-Reynolds number regions. The results obtained by the wall-function method and the low-Re modelling will be compared in order to evaluate these two approaches for treating

29

the wall regions. Finally, an effort will also be made to introduce compressibility in the analysis. For this purpose, the two-equation turbulence model of Launder and Spalding (Ref. 32) will be used to study a typical compressible flow problem such as the flow exhausting from a rectangular nozzle for which the corresponding laminar-flow analysis and solution procedure are already available.

3.   PERSONNEL

The research reviewed in this annual report represents
the efforts of the two principal investigators for task II
of the subject research project: Drs. K.N. Ghia and U. Ghia.
Also, Mr. G.A. Osswald and Mr. C.T. Shin, graduate research
assistants in the Aerospace Engineering Department, have
assisted in the research. Mr. Osswald's efforts are directed
primarily in the area of developing and employing direct
Poisson solvers in order to improve the efficiency of numerical
solutions of the Navier-Stokes equations. Mr. Shin's contri-
butions have been in the use of spline approximations in the
descretization of the velocity-pressure formulation of the
Navier-Stokes equations. These efforts of Mr. Osswald and
Mr. Shin are leading to their Master's degree theses which are
expected to be completed during the remainder of the contract
period.

4. REFERENCES

Note: Superscript asterisk (*) denotes that the work was per-
formed partially or wholly, under AFOSR sponsorship.

1. Ghia, U., Ghia, K.N. and Studerus, C.J., "Use of Surface-
    Oriented Coordinates in the Numerical Simulation of Flow
    in a Turbine Cascade," Lecture Notes in Physics, Vol. 59,
    1976.

2. Millsaps, K. and Pohlhausen, K., "Thermal Distribution in
    Jeffrey-Hamel Flows Between Non-Parallel Plane Walls,"
    J. Aero. Sci., March 1953, pp. 187-196.

3.* Sathyanarayana, K. and Ghia, U., "Numerical Simulation of
    Three-Dimensional Viscous Flow in Ducts of Arbitrarily
    Varying Cross Sections," submitted for presentation at
    Second International Conference on Mathematical Modelling,
    St. Louis, Missouri, July 1979.

4.* Ghia, U. and Sathyanarayana, K., "Analysis and Solution of
    Three-Dimensional Viscous Flow in Non-Rotating Ducts,"
    submitted for presentation at ASME Winter Annual Meeting,
    New York, December 1979.

5.* Ghia, U. and Sathyanarayana, K., "Three-Dimensional Viscous
    Flow in Joukowski Ducts," to be submitted for publication
    in Journal of Fluids Engineering, 1979.

6. Ghia, K.N., Hankey, W.L. and Hodge, J.K., "Use of Primitive
    Variables in the Solution of Incompressible Navier-Stokes
    Equations," AIAA Journal, Vol. 17, No. 3, March 1979,
    pp. 298-301.

7. Briley, W.R., "Numerical Method for Predicting Three-Dimensional
    Flow in Ducts," Journal of Computational Physics, Vol. 14,
    No. 1, 1974, pp. 8-28.

8. Ghia, U., Ghia, K.N. and Studerus, C.J., "Three-Dimensional
    Laminar Incompressible Flow in Straight Polar Ducts,"
    International Journal of Computers and Fluids, Vol. 5,
    No. 4, 1977.

9.* Ghia, K.N. and Sokhey, J.S., "Laminar Incompressible Viscous
    Flow in Curved Ducts of Regular Cross Sections," ASME
    Journal of Fluids Engineering, Vol. 99, December 1977,
    pp. 640-648.

10.* Ghia, U. and Goyal, R.K., "Three-Dimensional Viscous Incompressible
    Flow in Curved Polar Ducts," accepted for presentation at
    AIAA 12th Fluid and Plasma Dynamics Conference, Williamsburg,
    Virginia, July 1979.

11. Stewartson, K., "Multistructured Boundary Layers on Flat Plates and Related Bodies," _Advances in Applied Mechanics_, Vol. 14, 1974, pp. 145-239.

12. Ghia, U. and Davis, R.T., "Navier-Stokes Solutions for Flow Past a Class of Two-Dimensional Semi-Infinite Bodies," _AIAA Journal_, Vol. 12, No. 12, Dec. 1974, pp. 1659-1665.

13. Ghia, K.N., Ghia, U. and Tesch, W.A., "Evaluation of Several Approximate Models for Laminar Incompressible Separation by Comparison with Complete Navier-Stokes Solutions," _AGARD_ CP 168, May 1975, pp. 6.1 - 6.15.

14. Davis, R.T., "Numerical Solution of the Incompressible Navier-Stokes Equations for Two-Dimensional Flows at High Reynolds Number," _Proc. First Int. Conf. on Numerical Ship Hydrodynamics_, National Bureau of Standards, October 1975.

15. Rubin, S.G. and Khosla, P.K., "A Simplified Spline Solution Procedure," _Proc. Sixth Int. Conf. on Numerical Methods in Fluid Dynamics_, Georgian SSR, June 1978.

16. Smith, F.T., "Flow Through Constricted or Dilated Pipes and Channels," Parts 1 and 2, _Quarterly J. Mech. and Appl. Math._, Vol. 29, Pt. 3, 1976, pp. 343-376.

17.* Ghia, U., Ghia, K.N., Rubin, S.G. and Khosla, P.K., "Study of Incompressible Flow Separation Using Primitive Variables," accepted for presentation at _Symposium on Computers in Aerodynamics_, Polytechnic Institute of New York, New York, June 1979.

18. Rubin, S.G. and Khosla, P.K., "Higher-Order Numerical Solutions Using Cubic Splines," _AIAA Journal_, Vol. 14, No. 7, 1976, pp. 851-858.

19. Rubin, S.G. and Khosla, P.K., "Polynomial Interpolation Methods for Viscous Flow Calculations," _Journal of Computational Physics_, Vol. 24, No. 3, 1977, pp. 217-244.

20. Mikhail, A.G. and Ghia, K.N., "Simultaneous Solution of n Second-Order and (4-n) First-Order Ordinary Differential Equations; $n \leq 4$," to appear as Aerospace Engineering Report, University of Cincinnati, 1979.

21. Beam, R.M. and Warming, R.F., "An Implicit Factored Scheme for the Compressible Navier-Stokes Equations," _AIAA Paper 77-645_, Proceedings of AIAA 3rd Computational Fluid Dynamics Conference, Albuquerque, NM, July 1977.

22. Kondrat'ev, V.A., "Asymptotic of Solution of the Navier-Stokes Equation Near the Angular Point of the Boundary," _Journal of Applied Mathematics and Mechanics, USSR_, Vol. 31, 1967, pp. 125-129.

23. Ghia, K.N., Shin, C.T. and Ghia, U., "Use of Spline Approximations for Higher-Order Accurate Solutions of Navier-Stokes Equations in Primitive Variables," accepted for presentation at AIAA 4th Computational Fluid Dynamics Conference, Williamsburg, Virginia, July 1979.

24. Hockney, R.W., "The Potential Calculation and Some Applications," Methods in Computational Physics, Vol. 9, Academic Press, New York and London, 1969, pp. pp. 136-311.

25. Buneman, O., "A Compact Non-Iterative Poisson Solver," SUIPR Report No. 294, 1969.

26. Buzbee, B.L., Golub, G.H. and Nielson, C.W., "On Direct Methods for Solving Poisson's Equations," SIAM J. Numer. Anal., Vol. 7, 1970, pp. 627-656.

27. Roache, P.J., Computational Fluid Dynamics, Hermosa Publishers, Albuquerque, New Mexico, 1972.

28. Rubin, S.G. and Harris, J.E., "Numerical Studies of Incompressible Viscous Flow in a Driven Cavity," NASA SP-378, 1975.

29. Blottner, F.G., "Variable Grid Scheme Applied to Turbulent Boundary Layers," Computer Methods in Applied Mechanics and Engineering, Vol. 4, 1974, pp. 179-194.

30. Harlow, F.H. and Welch, J.E., "Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface," The Physics of Fluids, Vol. 8, No. 12, 1965, pp. 2182-2189.

31. Osswald, G.A. and Ghia, K.N., "Use of Direct Poisson Solvers in the Numerical Solution of Navier-Stokes Equations," submitted for presentation at Second International Conference on Mathematical Modelling, St. Louis, Missouri, July 1979.

32. Launder, B.E. and Spalding D.B., "Numerical Computation of Turbulent Flows," Computer Methods in Applied Mechanics and Engineering, Vol. 3, 1974, pp. 269-289.

33. Sokhey, J.S., Ghia, K.N. and Ghia, U., "Study of Turbulent Flow in Curved Ducts Using a Two-Equation Turbulence Model," Proceedings of First International Conference on Mathematical Modelling, St. Louis, Missouri, September 1977.

34. Jones, W.P. and Launder, B.E., "The Prediction of Laminarization with a Two-Equation Model of Turbulence," International Journal of Heat and Mass Transfer, Vol. 15, 1972, pp. 301-314.

34

TABLE 1

UNSTEADY INCOMPRESSIBLE NAVIER-STOKES SOLVERS INVESTIGATED

THEORETICAL FORM OF TRANSPORT EQS. EMPLOYED	VARIABLES EMPLOYED	NUMERICAL SOLUTION PROCEDURE FOR		ABBREVIATIONS USED
		TRANSPORT EQS.	POISSON EQS.	
Conservative	(ω,ψ)	ADI	Buneman	$CS(\omega,\psi)$ ADIB
Conservative	(ω,ψ)	ADI	Hockney	$CS(\omega,\psi)$ ADIH
Convective	(ω,ψ)	ADI	Buneman	$CV(\omega,\psi)$ ADIB
Convective	(ω,ψ)	ADI	Hockney	$CV(\omega,\psi)$ ADIH
Conservative	(u,v,p)	ADI	Buneman	$CS(u,v,p)$ ADIB
Conservative	(u,v,p)	ADI	Hockney	$CS(u,v,p)$ ADIH
Convective	(u,v,p)	ADI	Buneman	$CV(u,v,p)$ ADIB
Convective	(u,v,p)	ADI	Hockney	$CV(u,v,p)$ ADIH
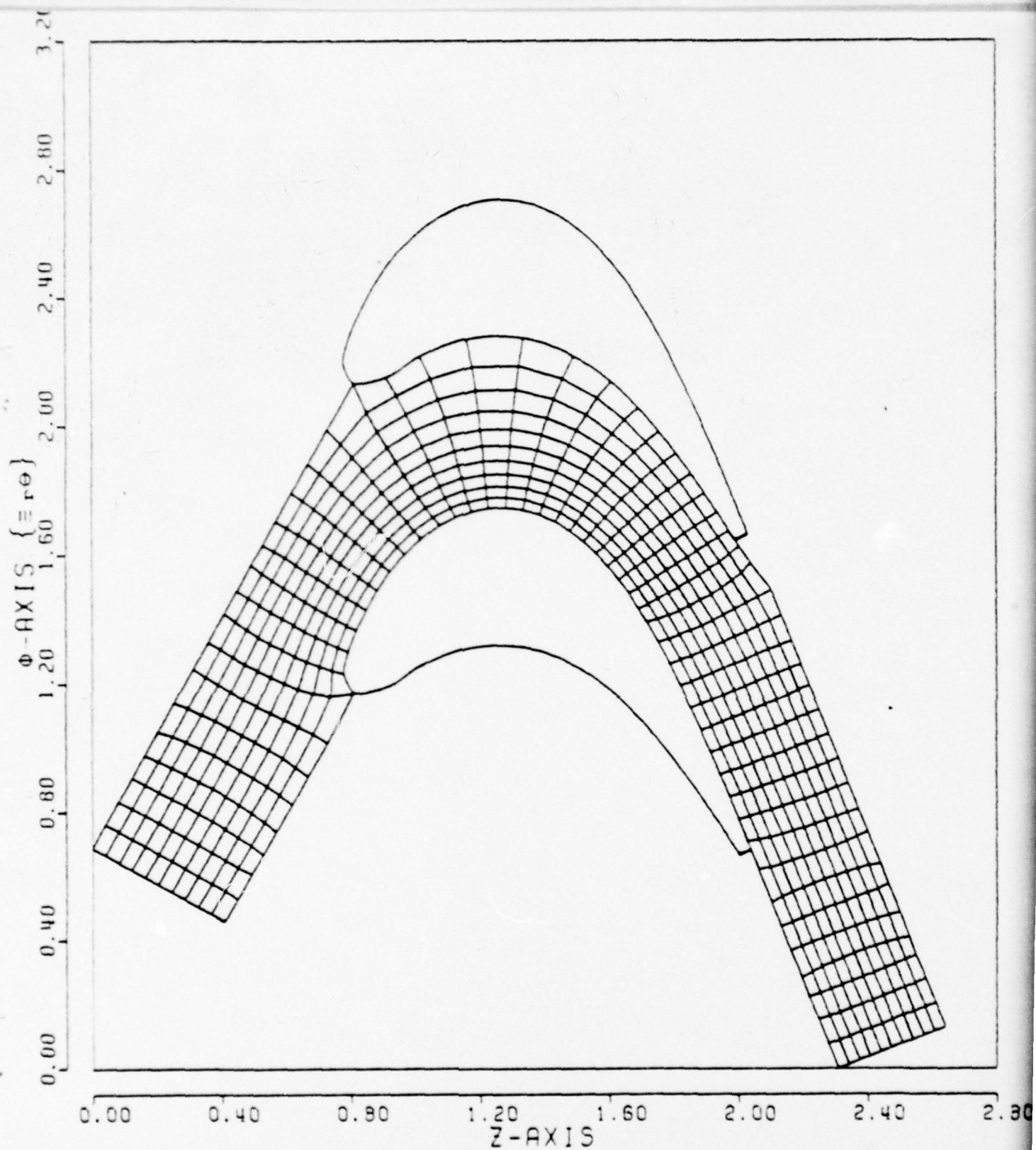
FIGURE 1. SURFACE-ORIENTED COORDINATES FOR VISCOUS FLOW THROUGH A TYPICAL ROTOR CASCADE.
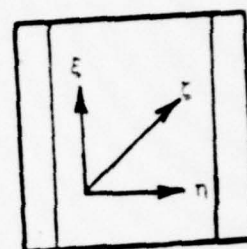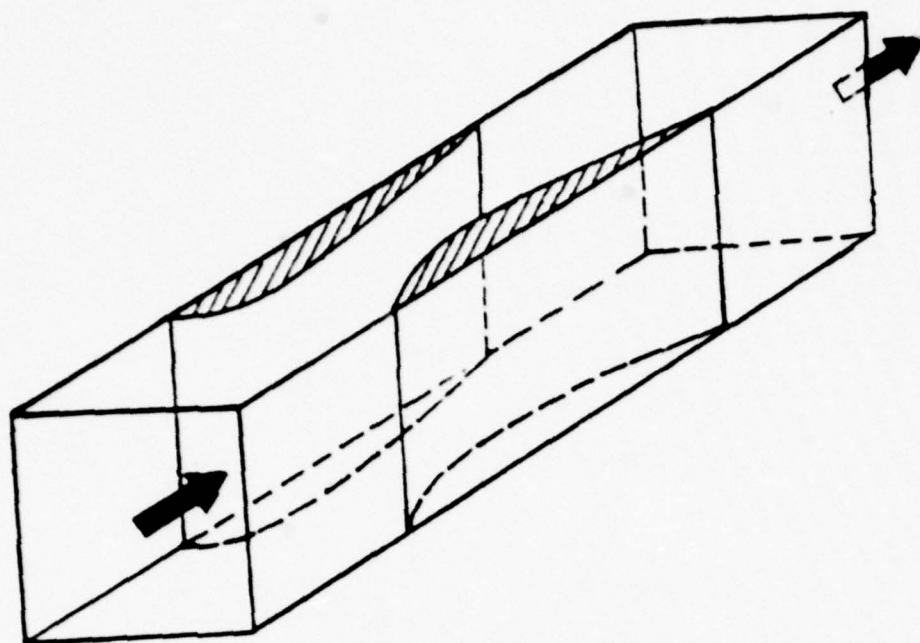
a. RECTANGULAR JOUKOWSKI DUCT
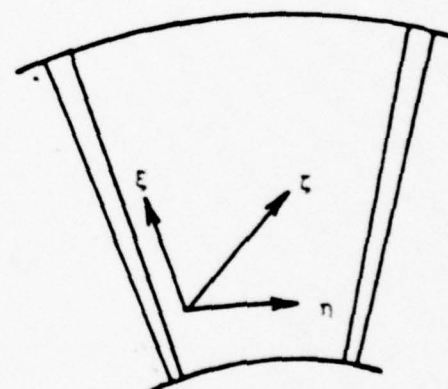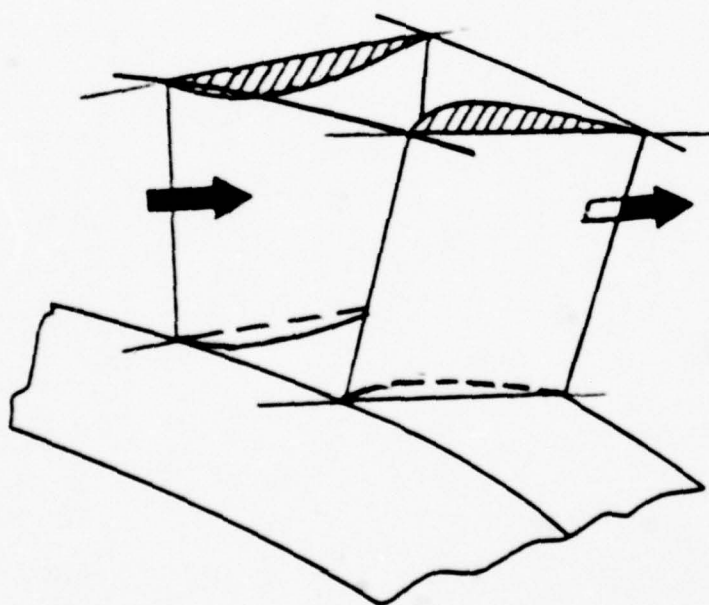
AXIAL VIEW

b. POLAR JOUKOWSKI DUCT

AXIAL VIEW

FIGURE 2. SCHEMATIC REPRESENTATION OF SIMPLIFIED CASCADE CHANNELS.

Area Ratio = 0.8; t_{max} = 0.0628, Re = 1000

C = 0.9, S = 0.628, Aspect Ratio = 1.0

$(w_{\mathfrak{z}}/w_1)$

$(z/D)/Re$

$(p_o-p)/0.5\rho w_1^2$

$(w_{\mathfrak{z}}/w_1)$

$(p_o-p_m)/0.5\rho w_1^2$

$\left[\dfrac{p_o-p_1}{0.5\rho w_1^2}\right]_{\mathfrak{z}}$

$\left[\dfrac{p_o-p_1}{0.5\rho w_1^2}\right]_{surface}$
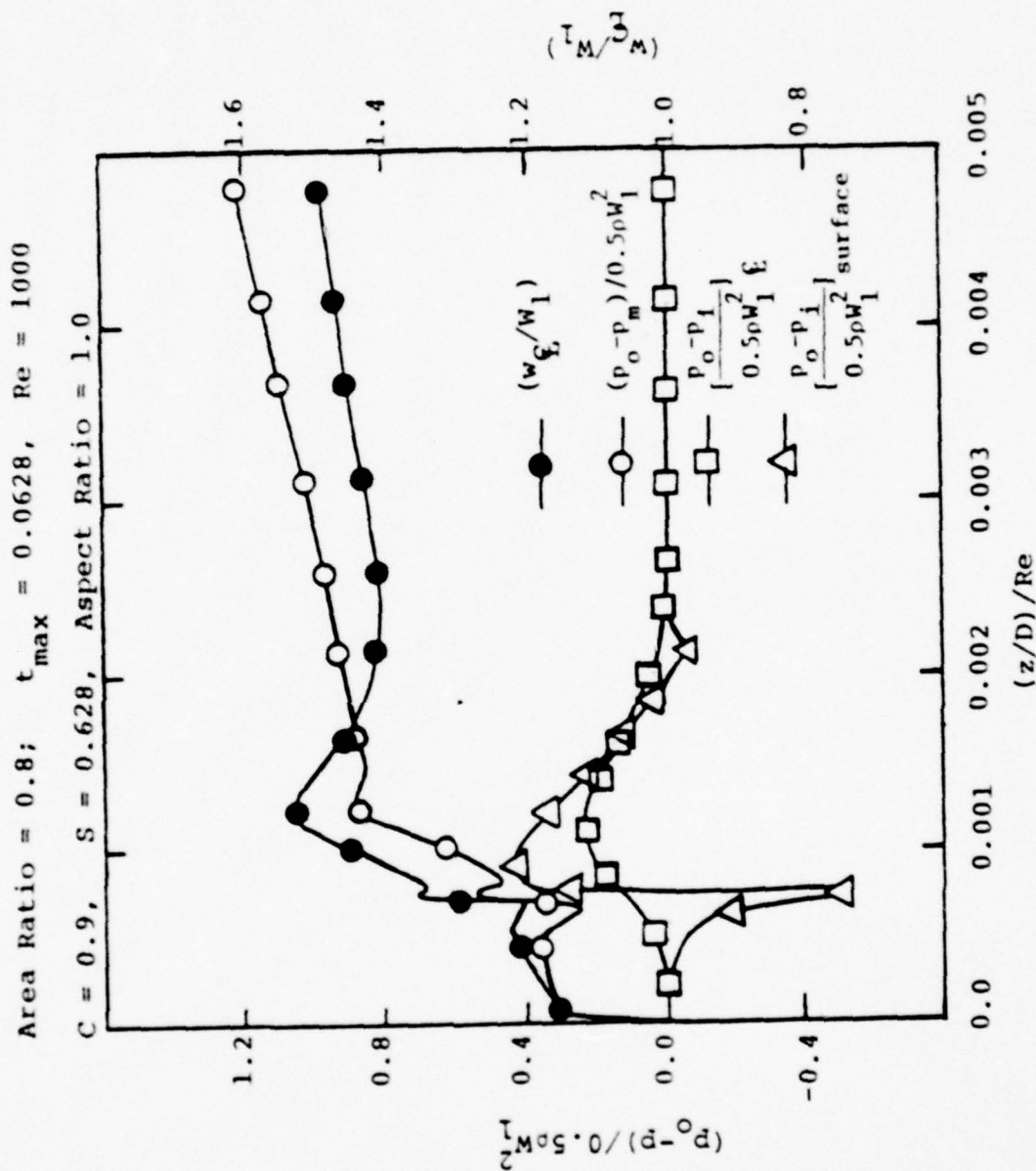
FIGURE 3. VARIATION OF CENTERLINE VELOCITY, MEAN-PRESSURE DROP AND INVISCID-PRESSURE DROP ALONG SURFACE AND CENTERLINE VS. NORMALIZED AXIAL DISTANCE FOR RECTANGULAR JOUKOWSKI DUCT.

38
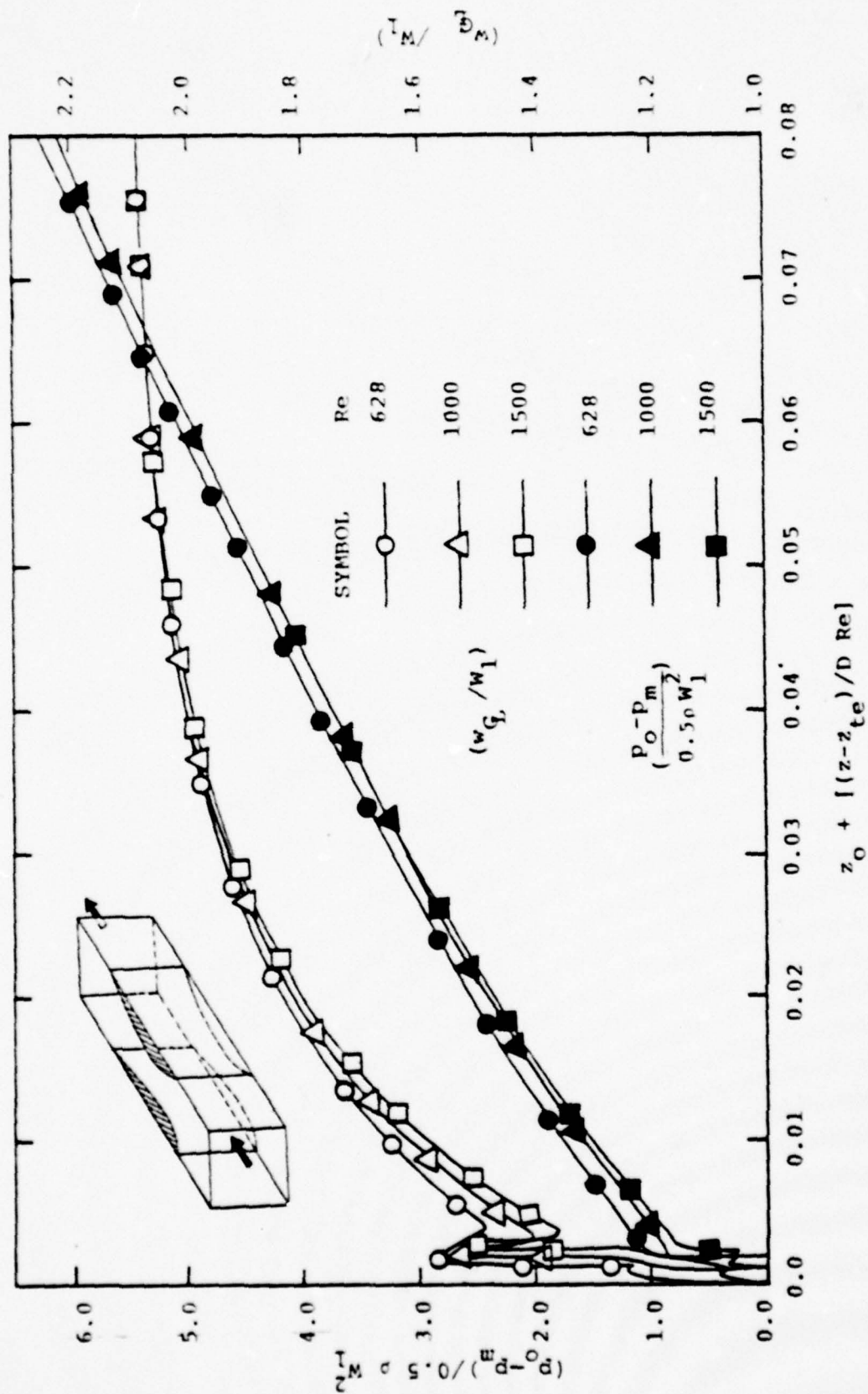
FIGURE 4. VARIATION OF CENTERLINE VELOCITY AND MEAN PRESSURE DROP VS. NORMALIZED AXIAL DISTANCE FOR RECTANGULAR JOUKOWSKI DUCTS WITH Re AS PARAMETER.

$$z_0 = (z_{te}/D \, Re_{ref}), \quad D = 0.628, \quad Re_{ref} = 628.$$

39

Aspect Ratio = 1.0,   Area Ratio = 0.8, t_{max} = 0.0628, Re = 628

b. PLANE 14

Scale
0.0 0.25 $\left(\dfrac{u}{W_1}\right)$ or $\left(\dfrac{v}{W_1}\right)$

a. PLANE 10

Scale
0.0 0.04 $\left(\dfrac{u}{W_1}\right)$ or $\left(\dfrac{v}{W_1}\right)$
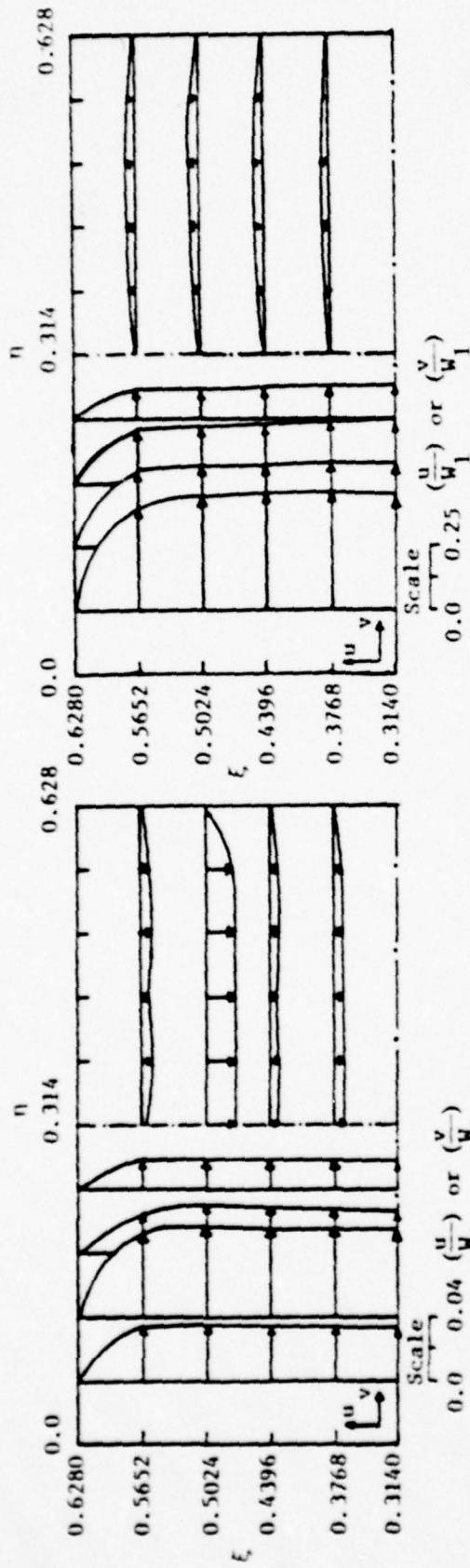
c. PLANE 20
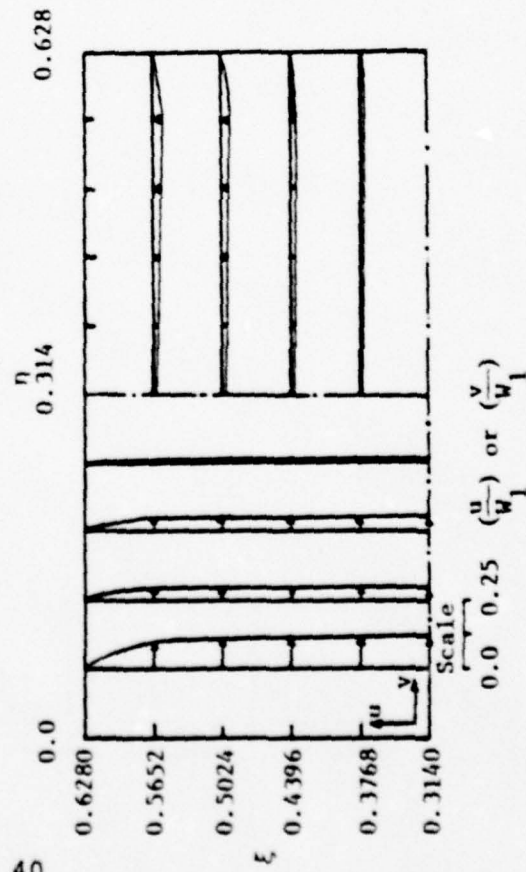
Scale
0.0 0.25 $\left(\dfrac{u}{W_1}\right)$ or $\left(\dfrac{v}{W_1}\right)$
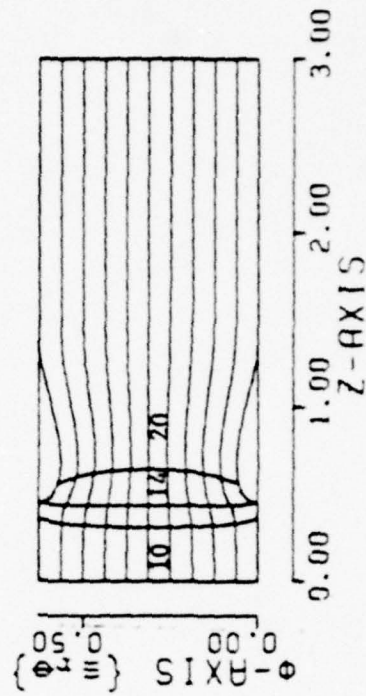
d. POSITION OF CROSS-SECTIONS 10, 14 AND 20.

Z-AXIS

Φ-AXIS $\{\equiv r\cdot\theta\}$

FIGURE 5A. DEVELOPMENT OF SECONDARY-VELOCITY PROFILES; PLANE NOS. 10, 14, 20.

40

ASPECT RATIO = 1.0, AREA RATIO = 0.8, t_{max} = 0.0628, Re = 628
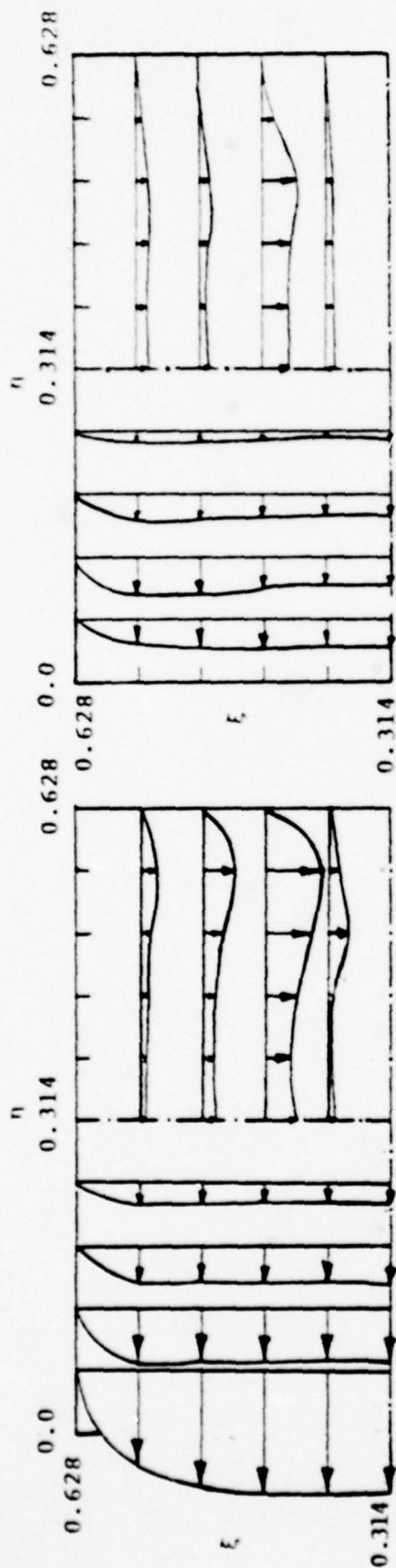
a. PLANE 27

b. PLANE 39

c. PLANE 70

d. POSITION OF CROSS SECTIONS 27, 39, and 70

FIGURE 5B. DEVELOPMENT OF SECONDARY-VELOCITY PROFILES; PLANE NOS. 27, 39, 70.

Scale
0.0 0.04 $\left(\frac{u}{W_1}\right)$ or $\left(\frac{v}{W_1}\right)$

Scale
0.0 0.04 $\left(\frac{u}{W_1}\right)$ or $\left(\frac{v}{W_1}\right)$

Scale
0.0 0.004

Z-AXIS

φ-AXIS { ≡ r-θ }

41
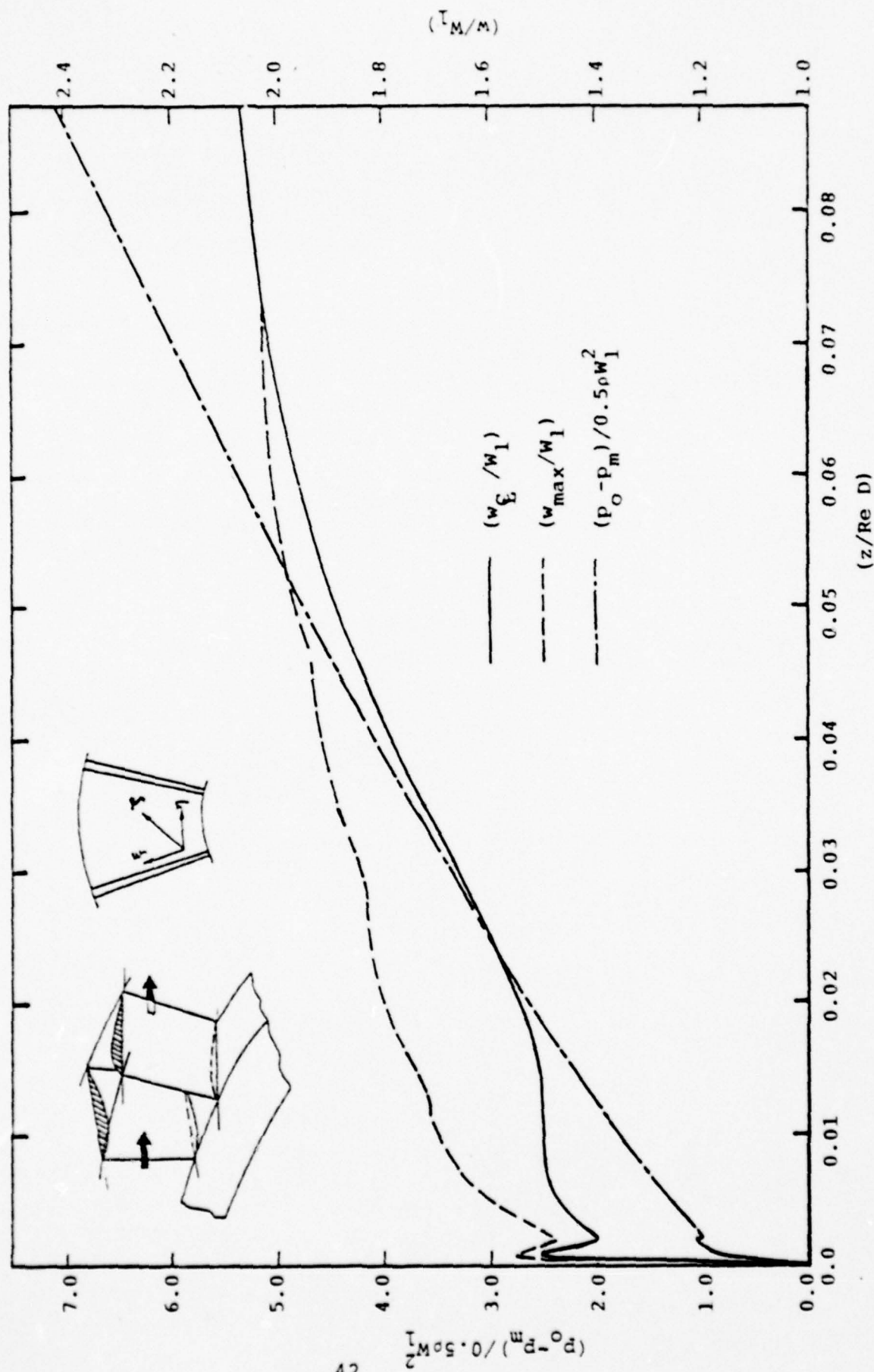
FIGURE 6. VARIATION OF CENTERLINE VELOCITY, MAXIMUM VELOCITY AND MEAN-VISCOUS PRESSURE DROP VS. NORMALIZED AXIAL DISTANCE FOR POLAR JOUKOWSKI DUCT.

42
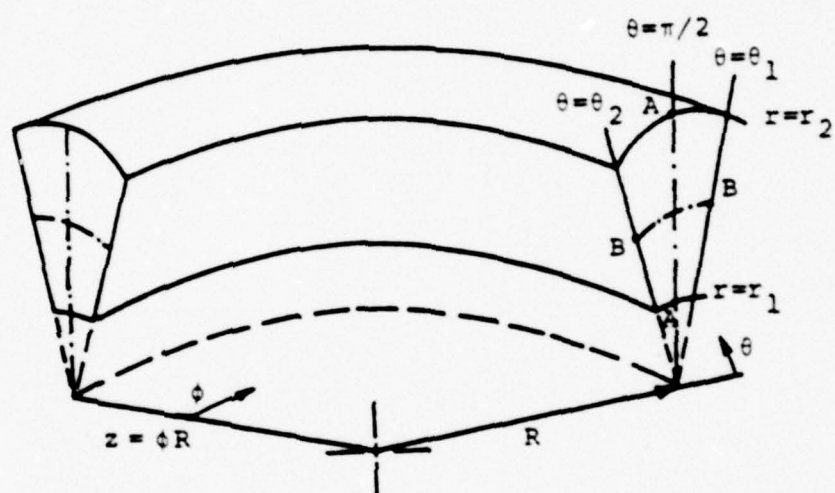
FIG. 7. CURVED DUCT GEOMETRY AND
COORDINATE SYSTEM.

43

1.8

AR = 1.0

CURVE	K	R_e	R
A	55	206	14
B	100	1000	100

1.7

1.6

A

1.5   10

B

W_{CL} B

1.4   8

1.3   A   6

1.2 $-P_m$

4

1.1   CURVED POLAR DUCT

△ ▲ CURVED RECTANGULAR
       DUCT   2

1.0
0   0.02 Z/R_e 0.06   0.08   0.1
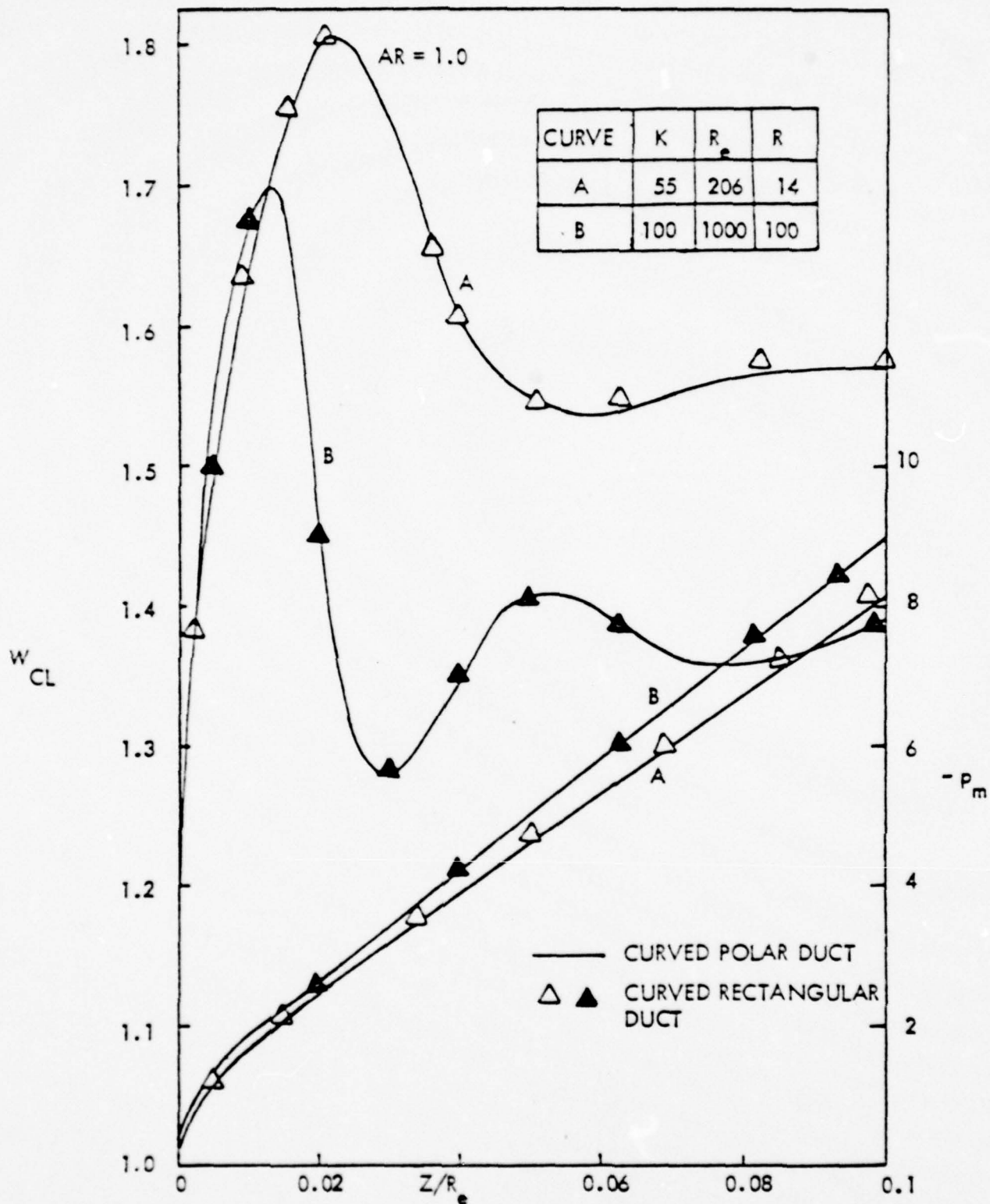
FIGURE 8.   COMPARISON OF STREAM WISE VARIATION OF CENTER
             LINE AXIAL VELOCITY AND MEAN VISCOUS PRESSURE
             FOR CURVED POLAR AND CURVED SQUARE DUCT

44
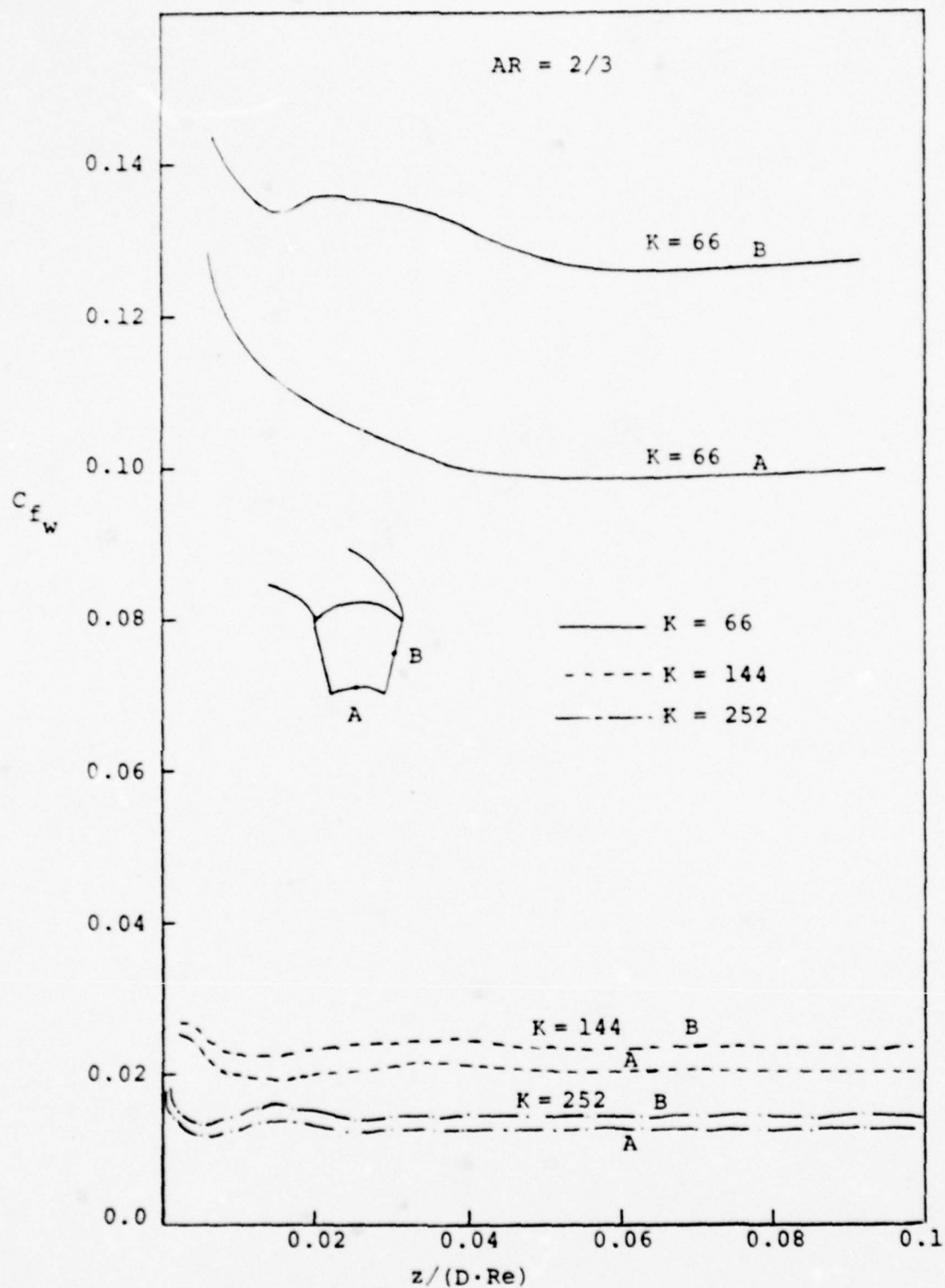
FIG. 9. EFFECT OF DEAN NUMBER ON SKIN-FRICTION IN
STREAMWISE DIRECTION IN A CURVED POLAR DUCT.

45
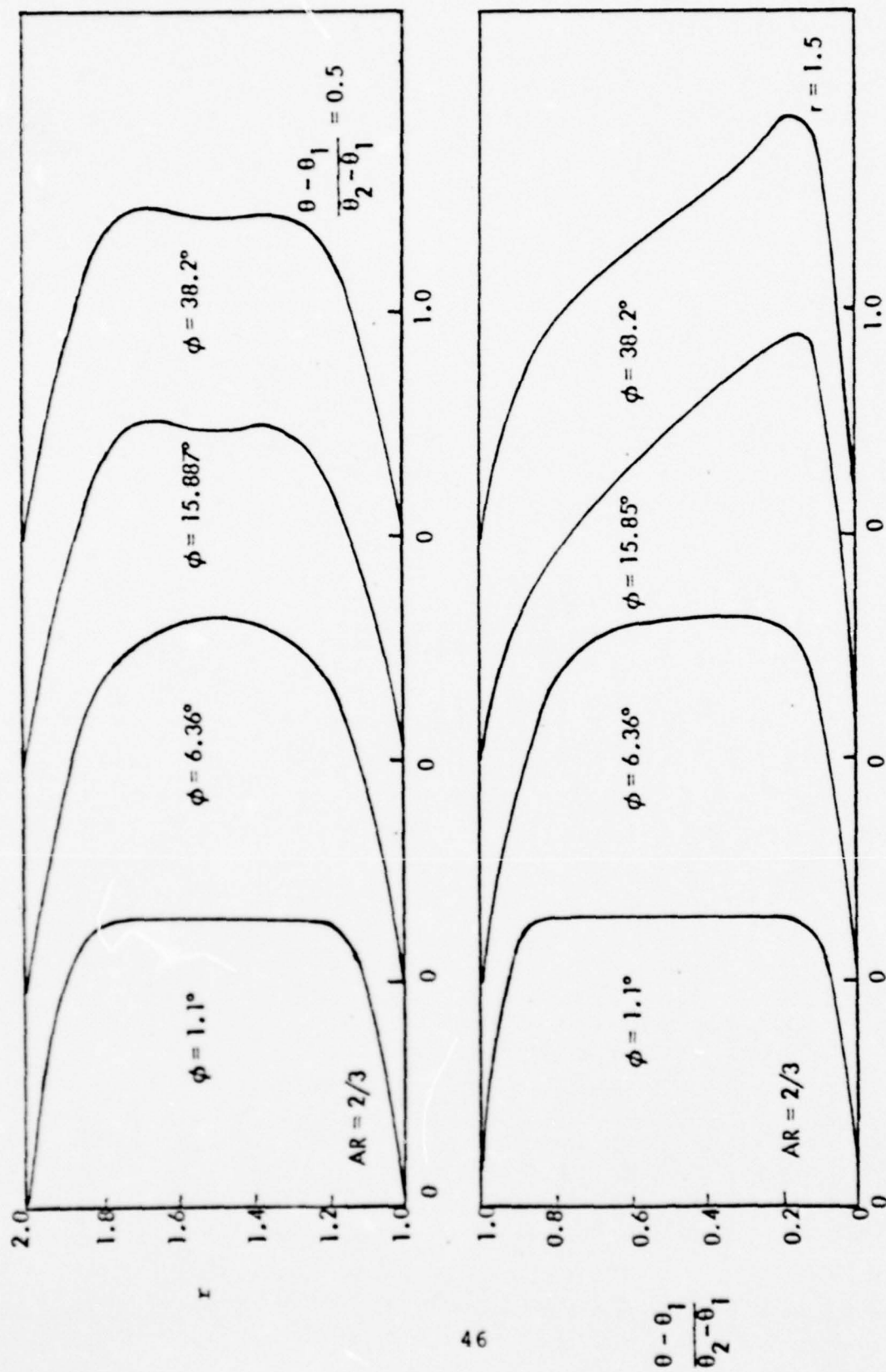
FIGURE 10a. DEVELOPMENT OF AXIAL VELOCITY PROFILES IN A CURVED POLAR–DUCT
K = 100, R_e = 937.2, R = 83.3 (15 × 15 NON–UNIFORM GRID)
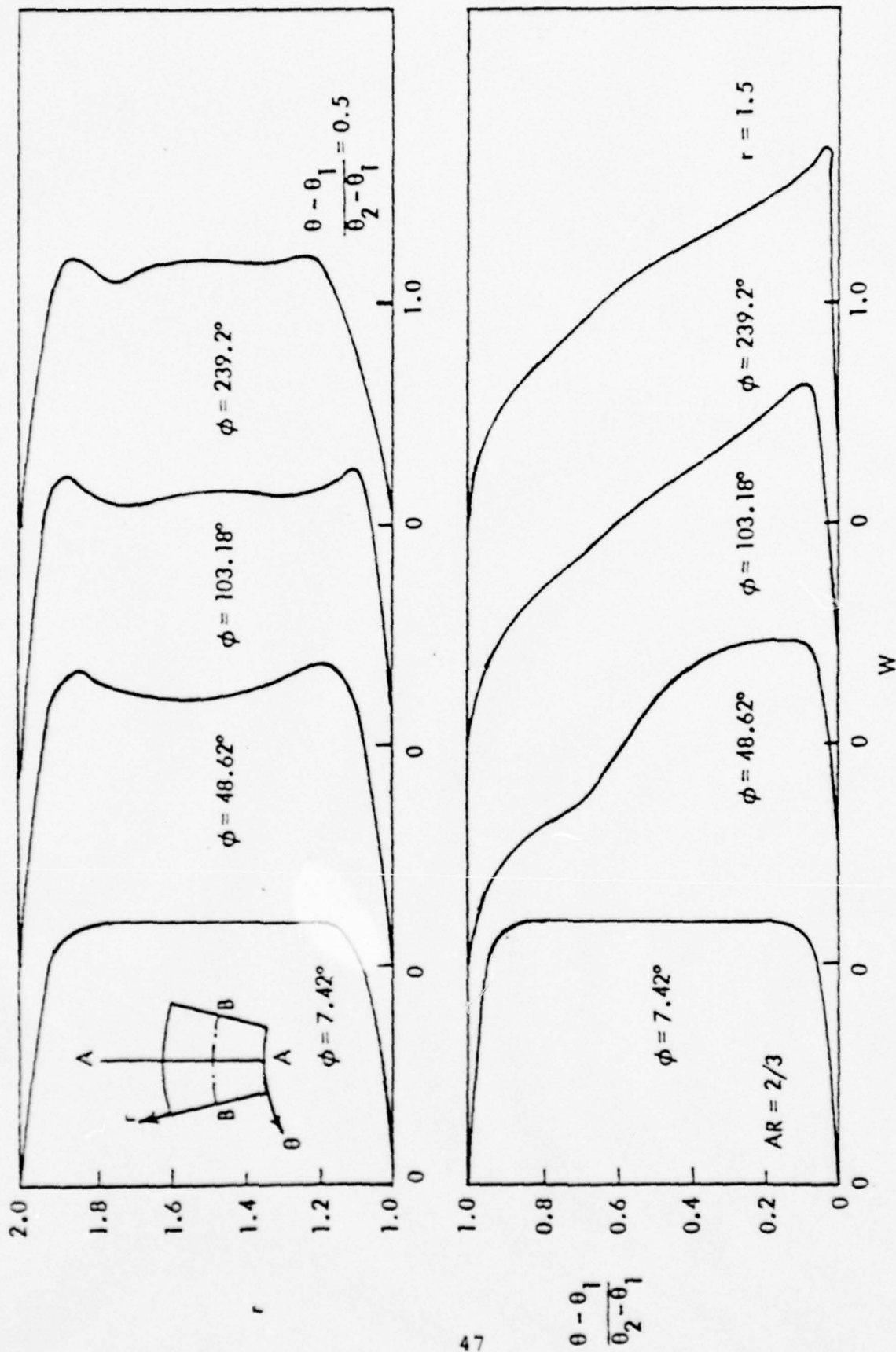
FIGURE 10b. DEVELOPMENT OF AXIAL VELOCITY PROFILES IN A CURVED POLAR DUCT
K = 630, R_e = 2520, R = 16, AR = 2/3
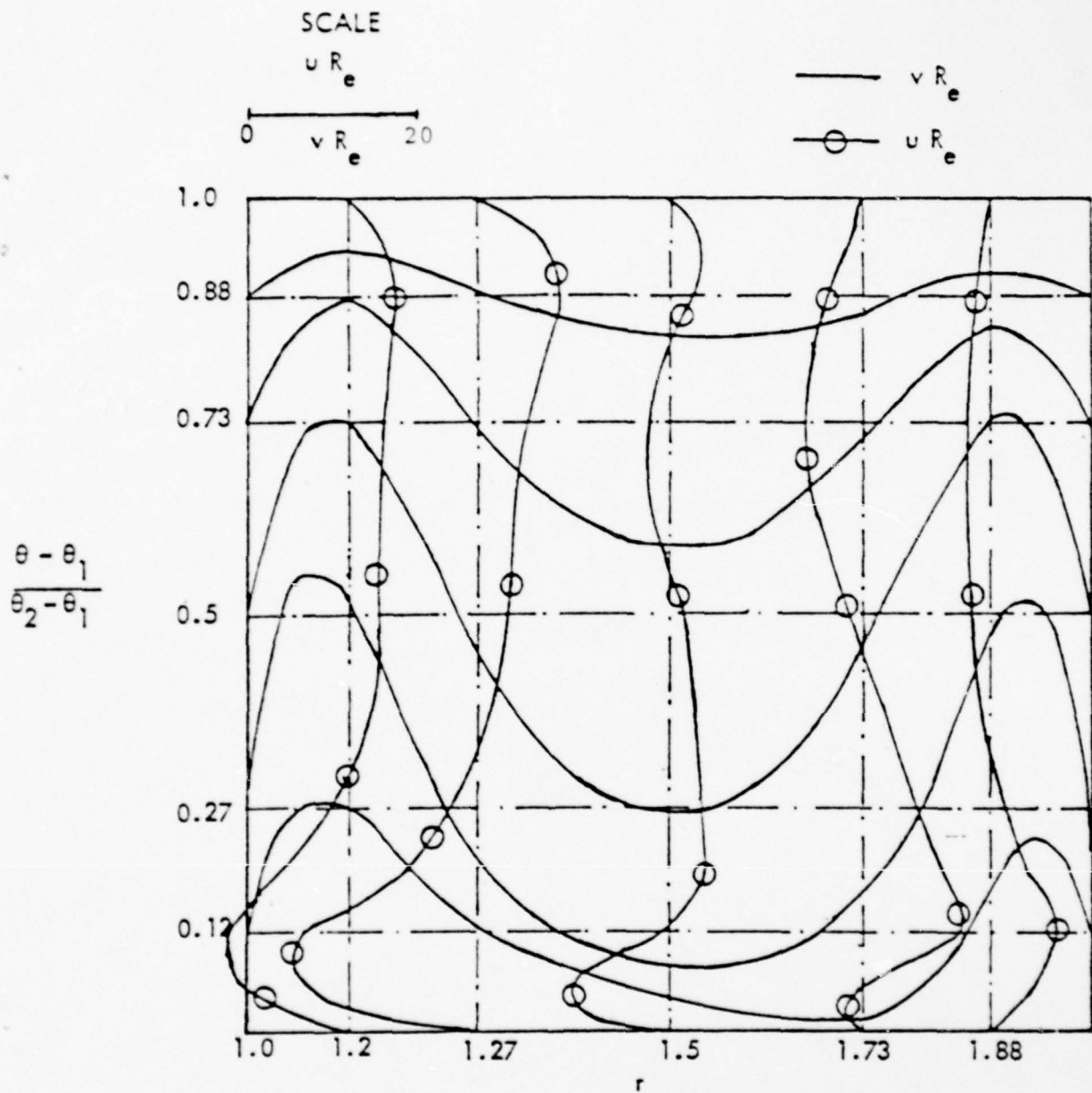
FIGURE 11a. FULLY DEVELOPED SECONDARY VELOCITY PROFILES
FOR A CURVED POLAR DUCT,
K = 100, R/D = 83.3, R_e = 913, AR = 2/3

48

SCALE

$u \cdot R_e$

0 ├────────────────┤ 320
$v \cdot R_e$

──────── $v \, R_e$

──○── $u \, R_e$

$\dfrac{\theta - \theta_1}{\theta_2 - \theta_1}$

0.89

0.74

0.5

0.26

0.11

1.0 1.1   1.26   1.5   2.0

r
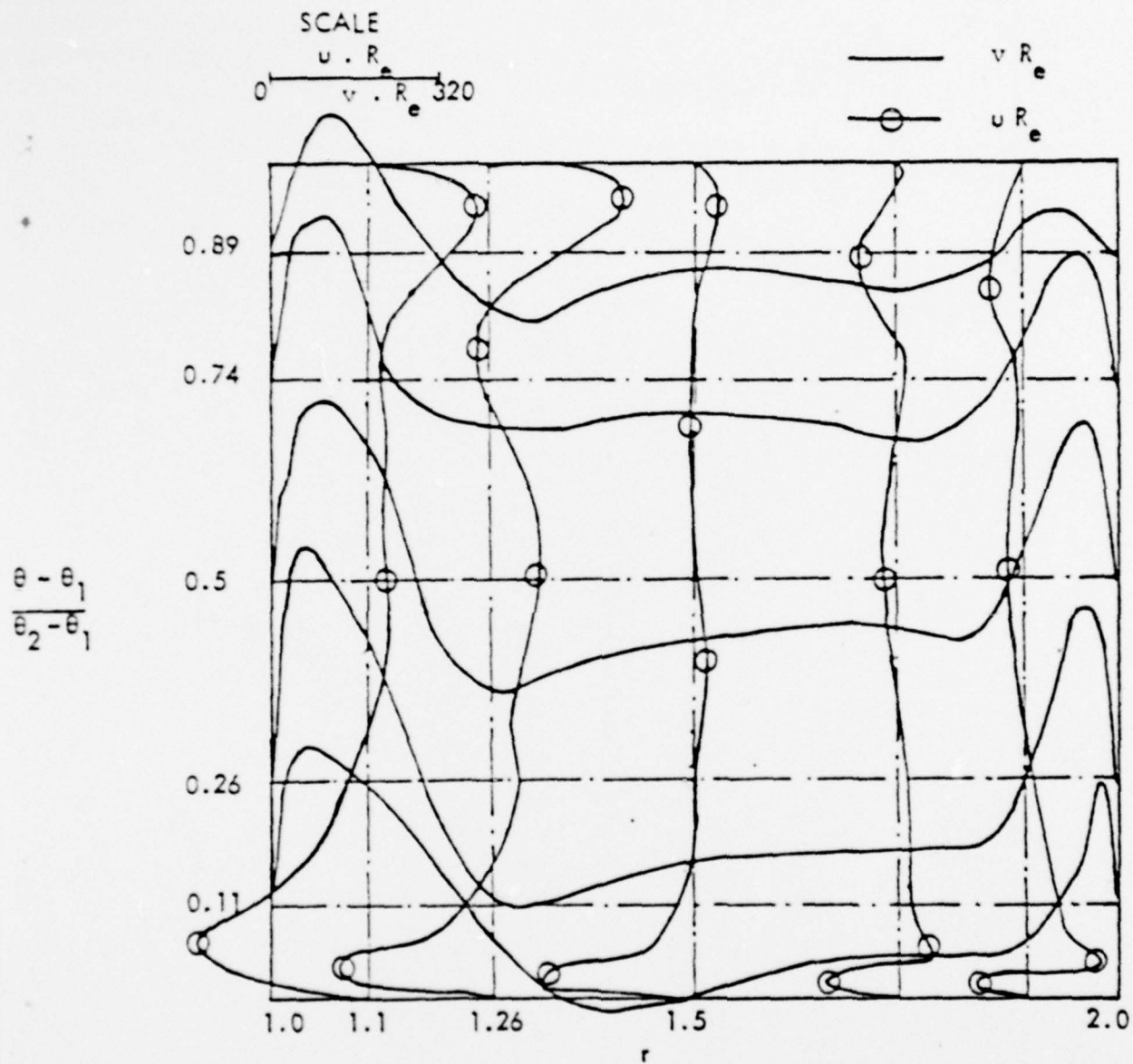
FIGURE 11b. FULLY DEVELOPED SECONDARY FLOW VELOCITY PROFILES
IN A CURVED POLAR DUCT K = 630, R = 16, AR = 2/3

49

Y1=0.200 L1=-3.000 L2=-3.030 d=0.644 S=0.285 A=2.000
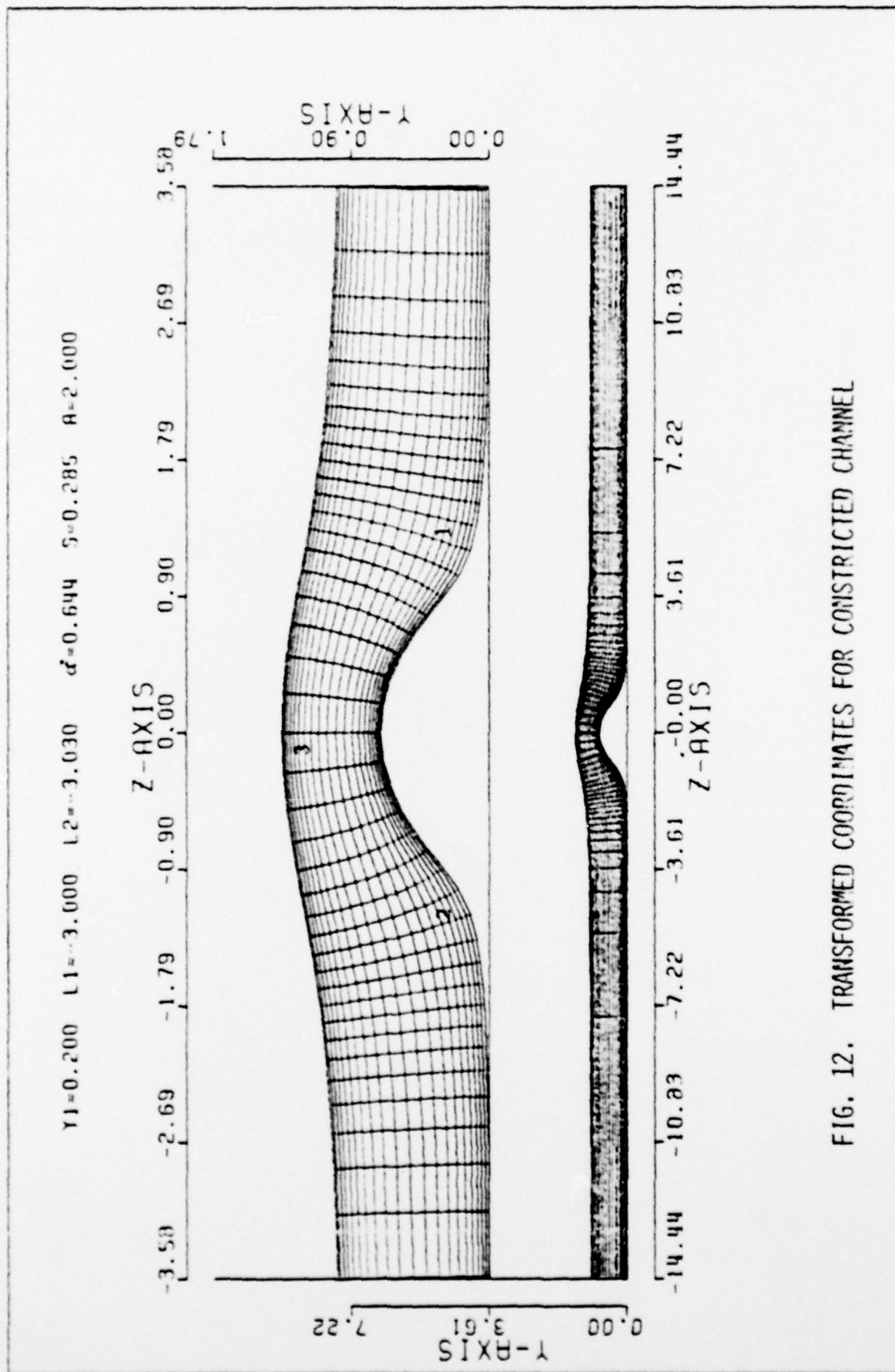
FIG. 12. TRANSFORMED COORDINATES FOR CONSTRICTED CHANNEL

Y1=0.200   L1=-3.000   L2=-3.030   α=0.000   S=0.285   A=2.000
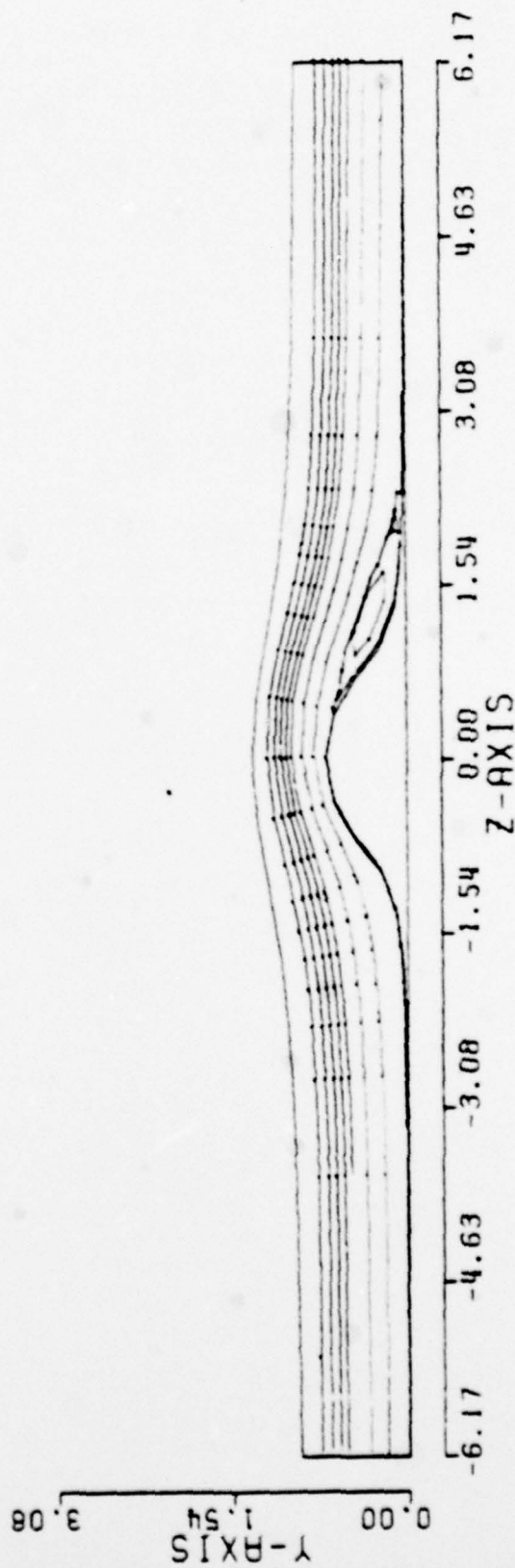
FIG. 13.   STREAMLINES FOR SEPARATED FLOW IN CONSTRICTED CHANNEL.
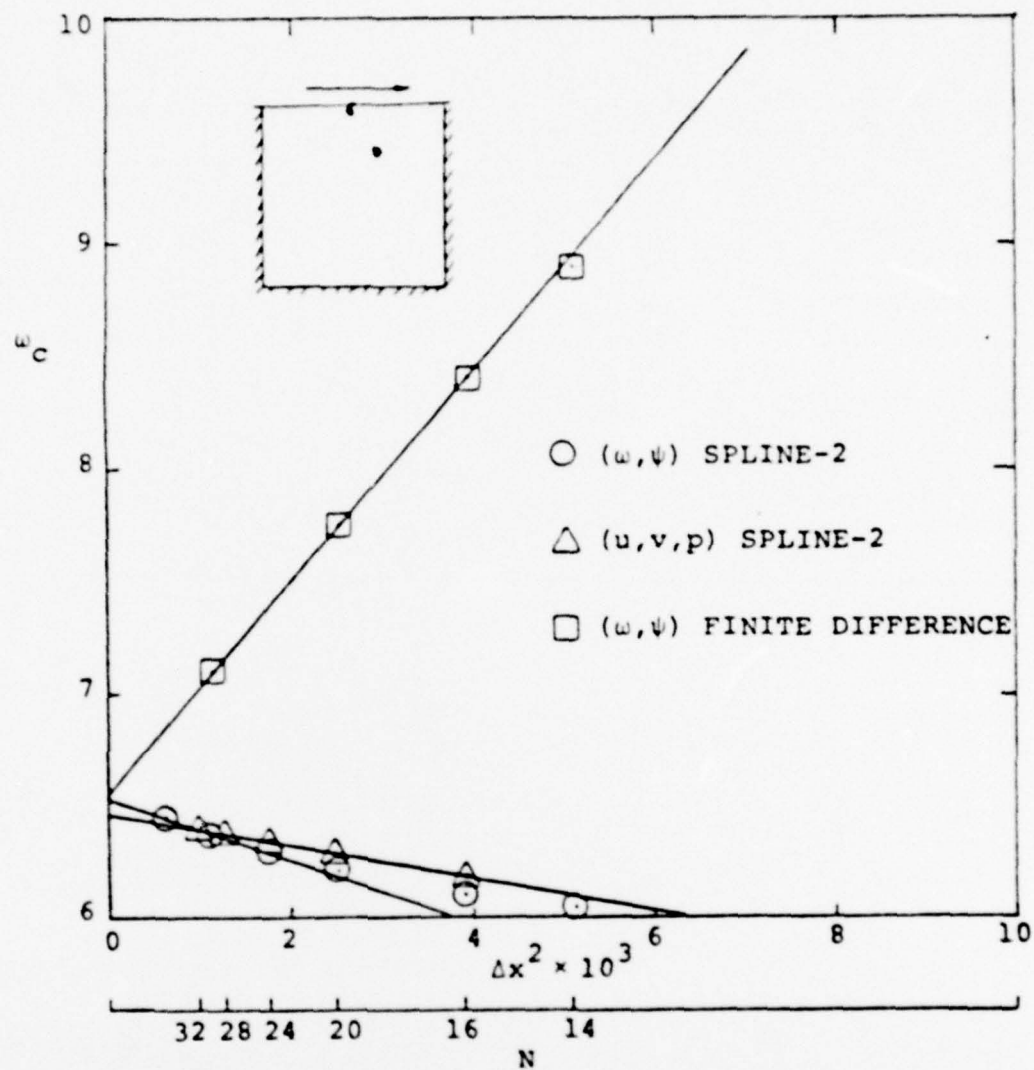
FIG. 14. ACCURACY OF SPLINE-2 SOLUTIONS AND FINITE-
DIFFERENCE SOLUTIONS FOR VORTICITY AT CENTER
OF MOVING WALL OF DRIVEN CAVITY.

52
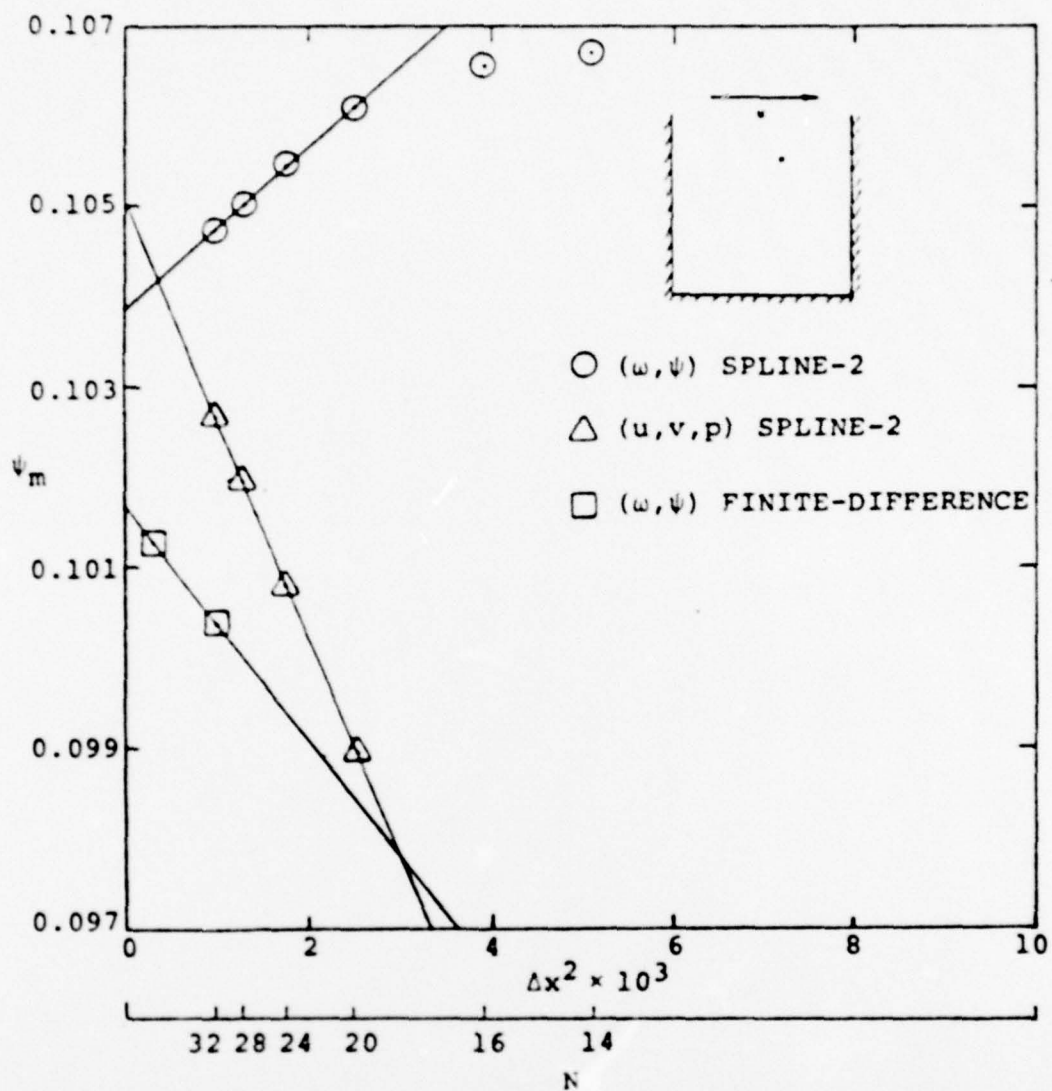
FIG. 15. ACCURACY OF SPLINE-2 SOLUTIONS AND FINITE-DIFFERENCE
SOLUTIONS FOR STREAM FUNCTION AT VORTEX CENTER
OF DRIVEN CAVITY.
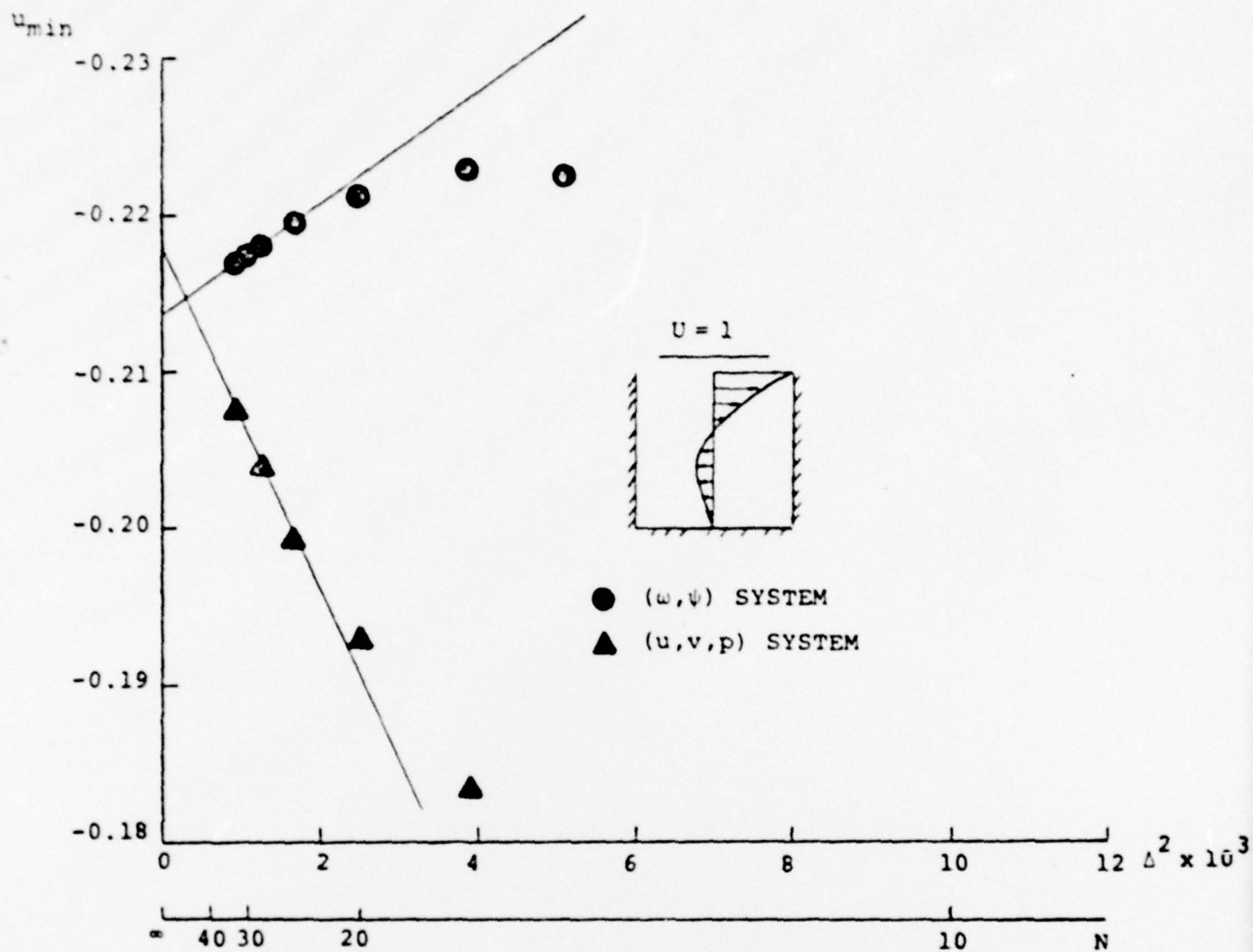
FIG. 16. ACCURACY OF (ω,ψ) SPLINE AND (u,v,p) SPLINE SOLUTIONS FOR u_{min} AT CAVITY CENTERLINE.
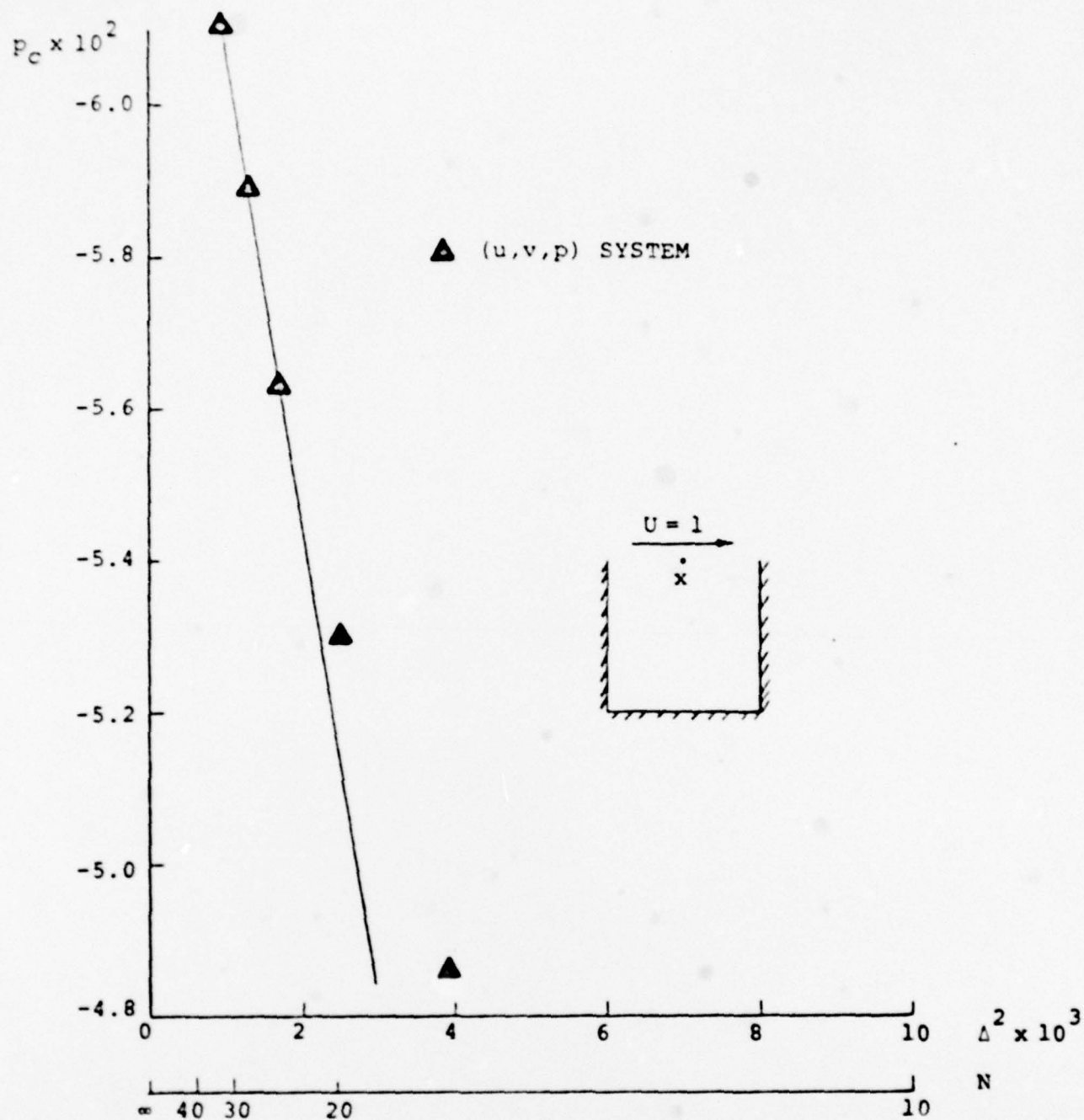
FIG. 17. STEP-SIZE STUDY FOR PRESSURE AT CENTER OF MOVING WALL
OF DRIVEN CAVITY.

55

RE=1000.   UNIFORM GRID (33X 33)   U-V-P
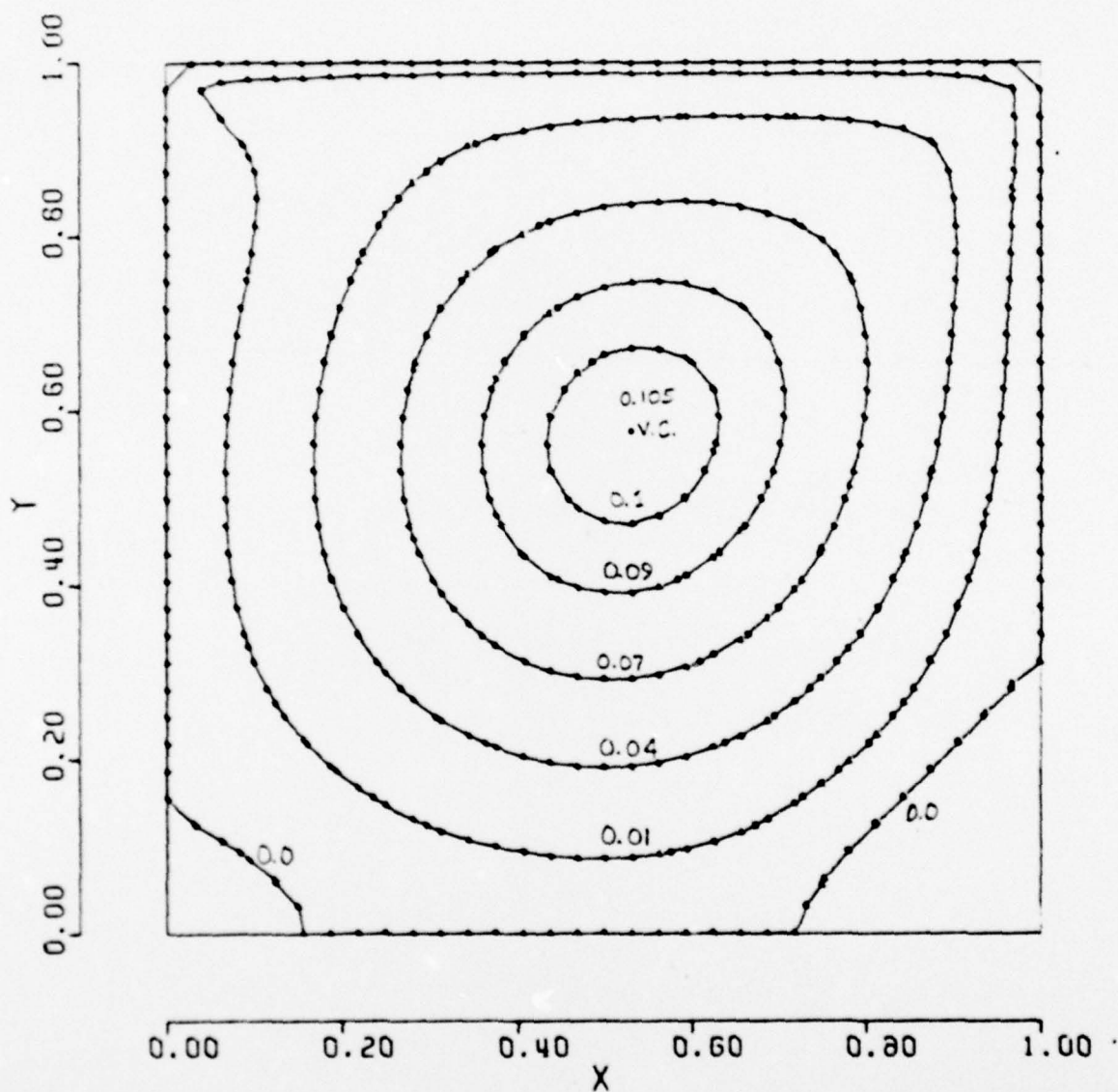
FIG. 18a. STREAMLINE CONTOURS FOR DRIVEN-CAVITY FLOW;
RE = 1000.

56

RE=1000.   UNIFORM GRID (33X 33)   u-v-r
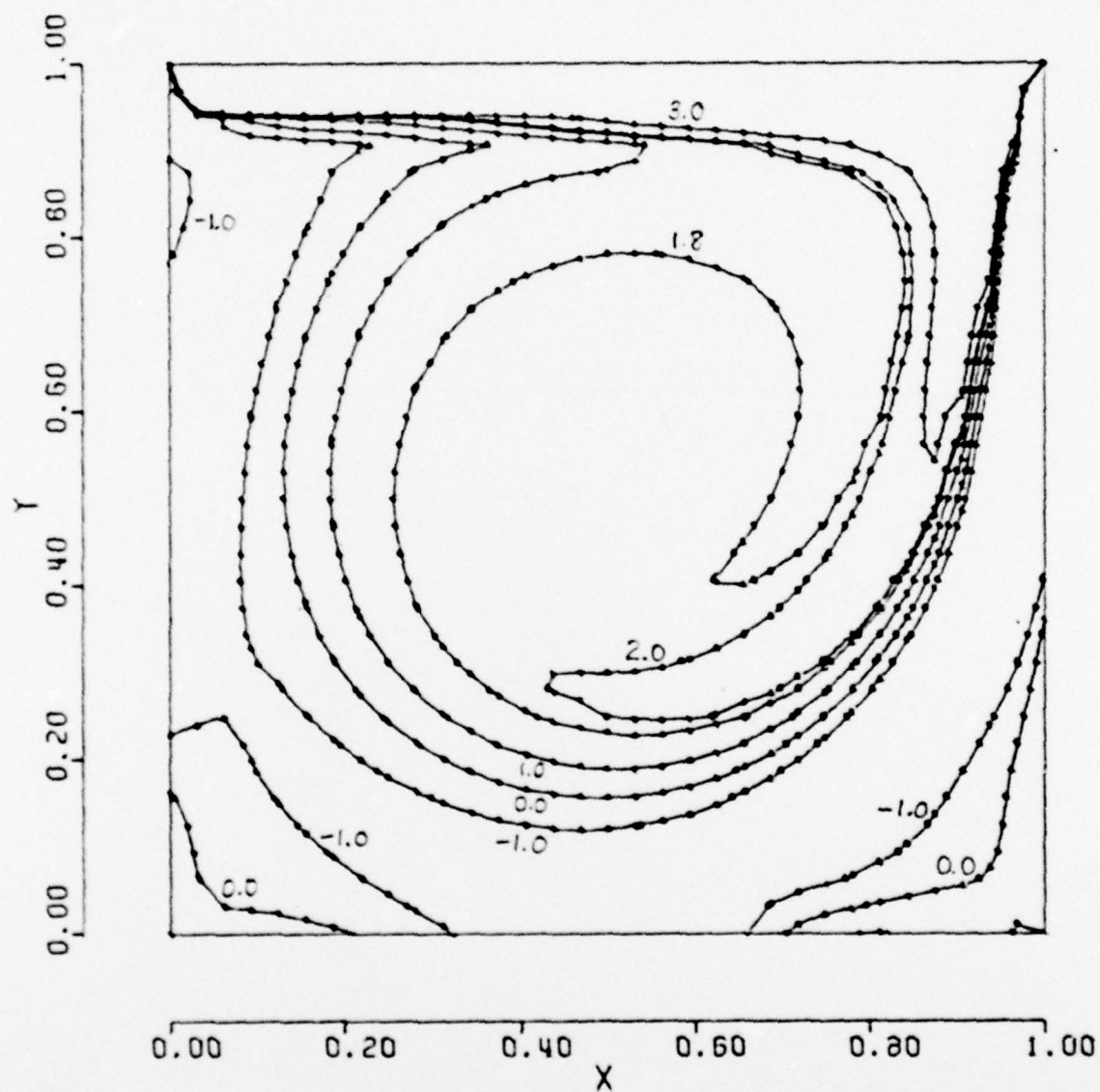
FIG. 18b. VORTICITY CONTOURS FOR DRIVEN-CAVITY FLOW;
           Re = 1000.

57

RE=1000.   UNIFORM GRID (33X 33)   U-V-P
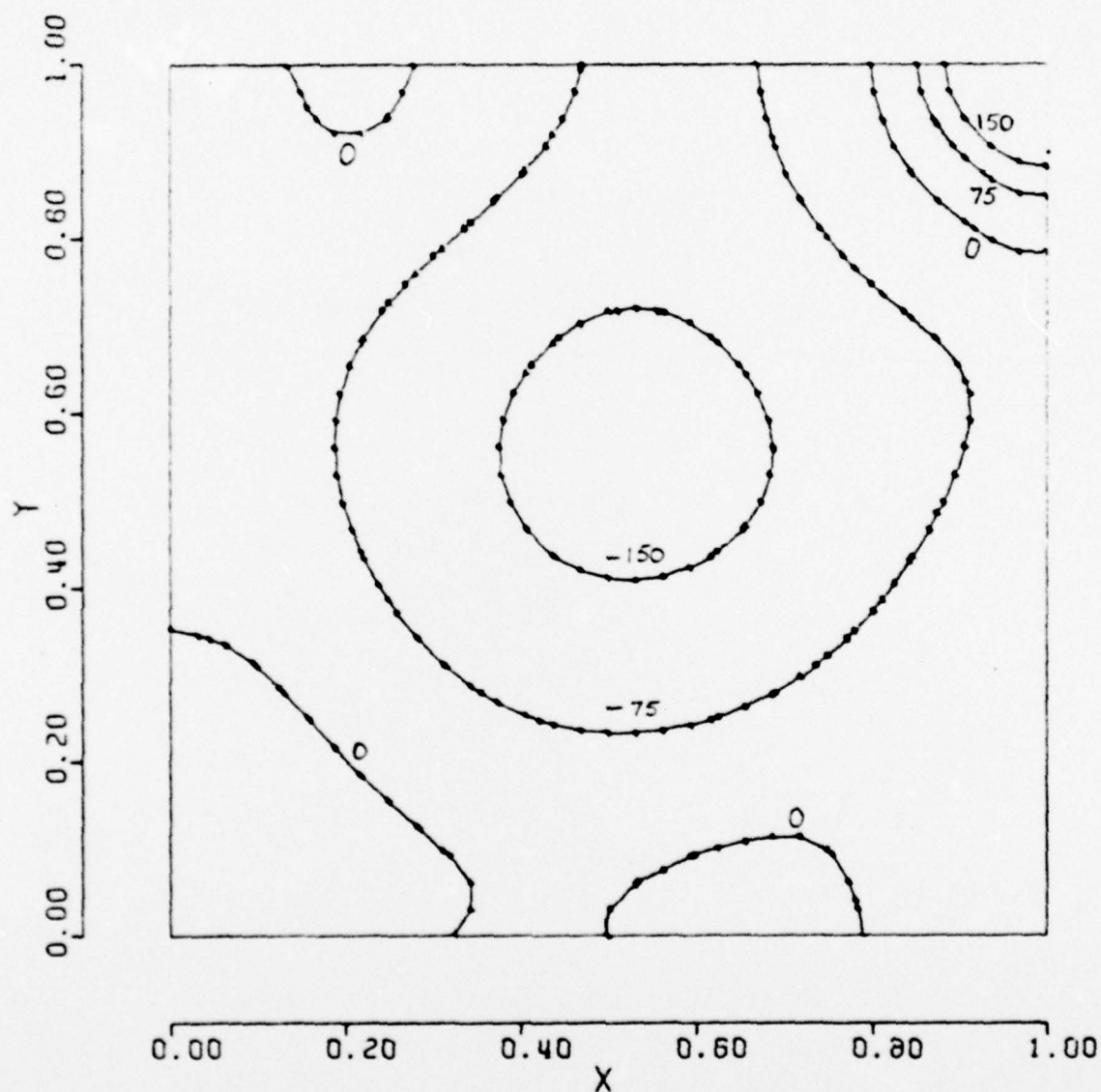
FIG. 18c.   STATIC PRESSURE CONTOURS FOR CAVITY FLOW;
          Re = 1000.
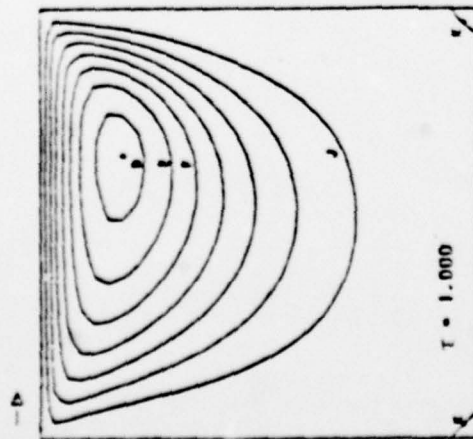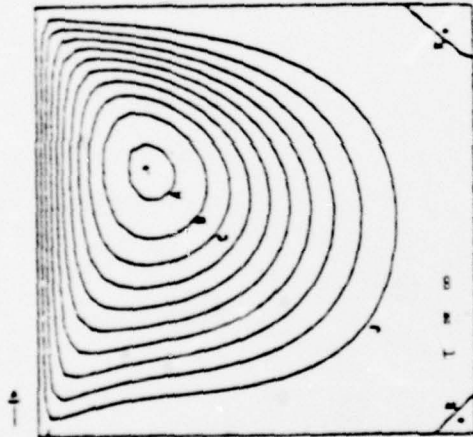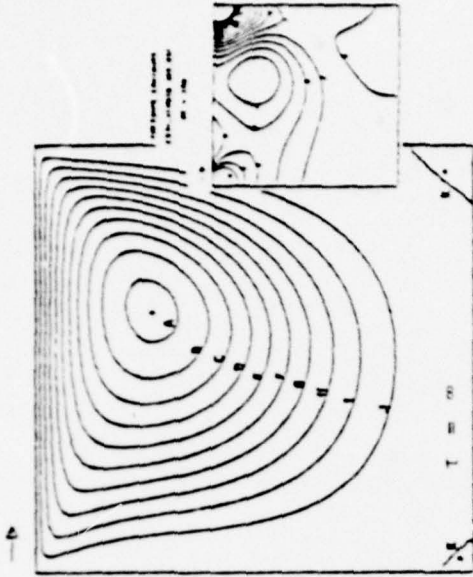
58

A = – 0.10   G = – 0.04
B = – 0.09   H = – 0.03
C = – 0.08   I = – 0.02
D = – 0.07   J = – 0.01
E = – 0.06   K   0.00
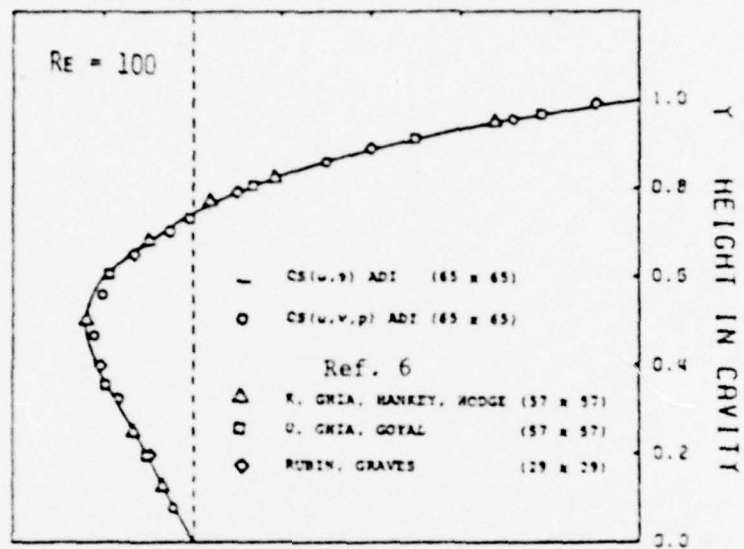F = – 0.05
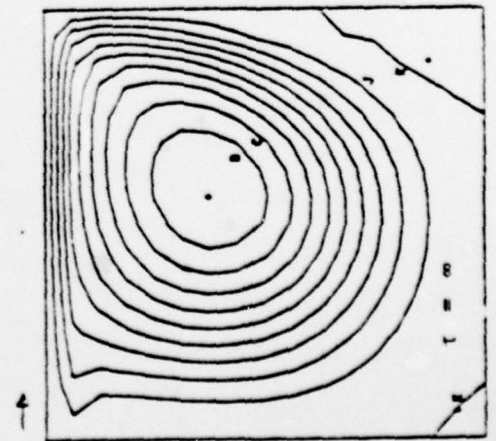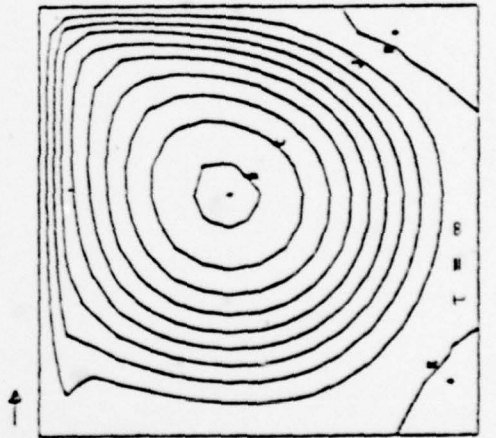
FIG. 19. STREAM-FUNCTION CONTOURS
FOR DRIVEN-CAVITY FLOW.

FIG. 20. COMPARISON OF CURRENT WORK WITH THAT OF
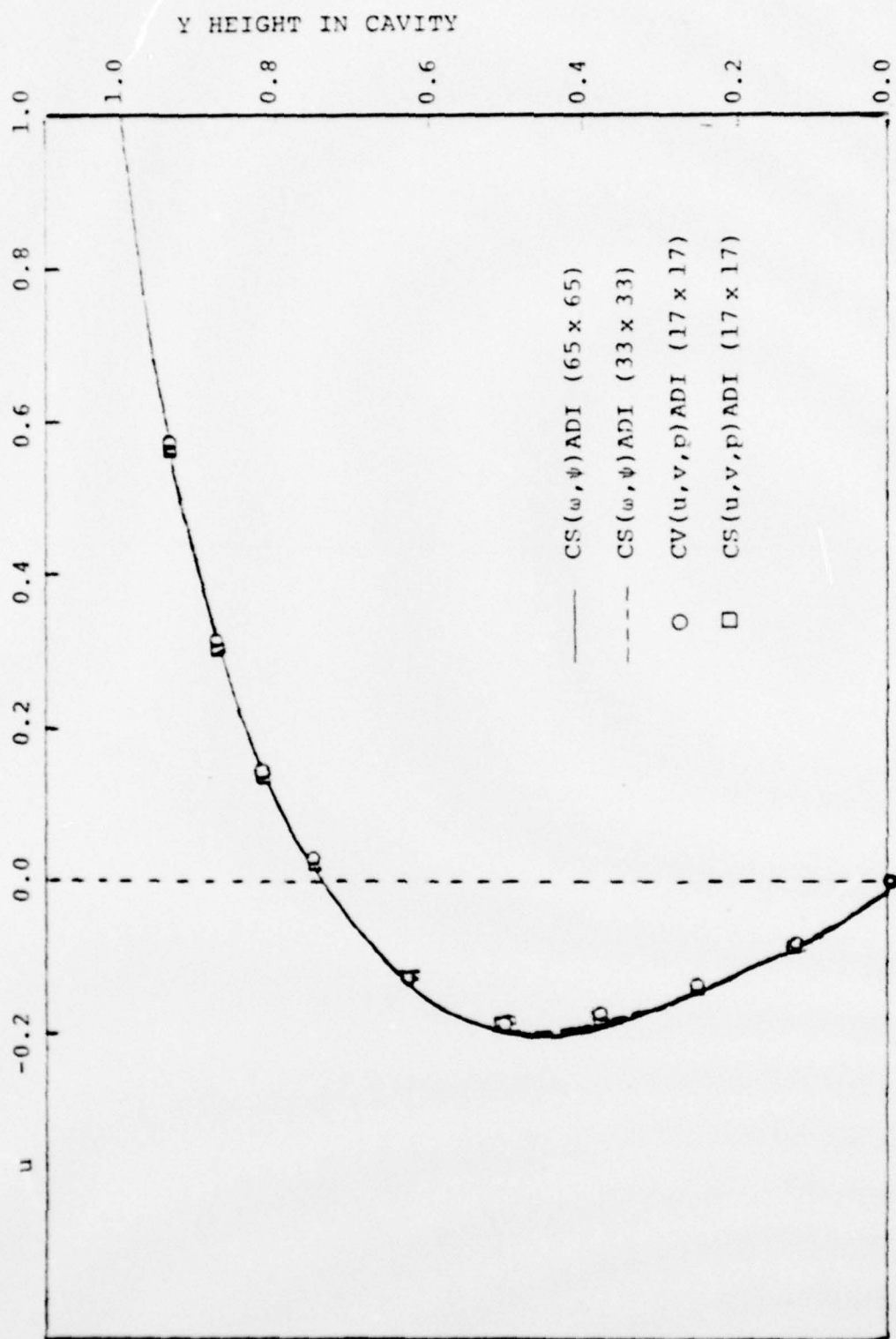PREVIOUS INVESTIGATORS FOR DRIVEN CAVITY.

60

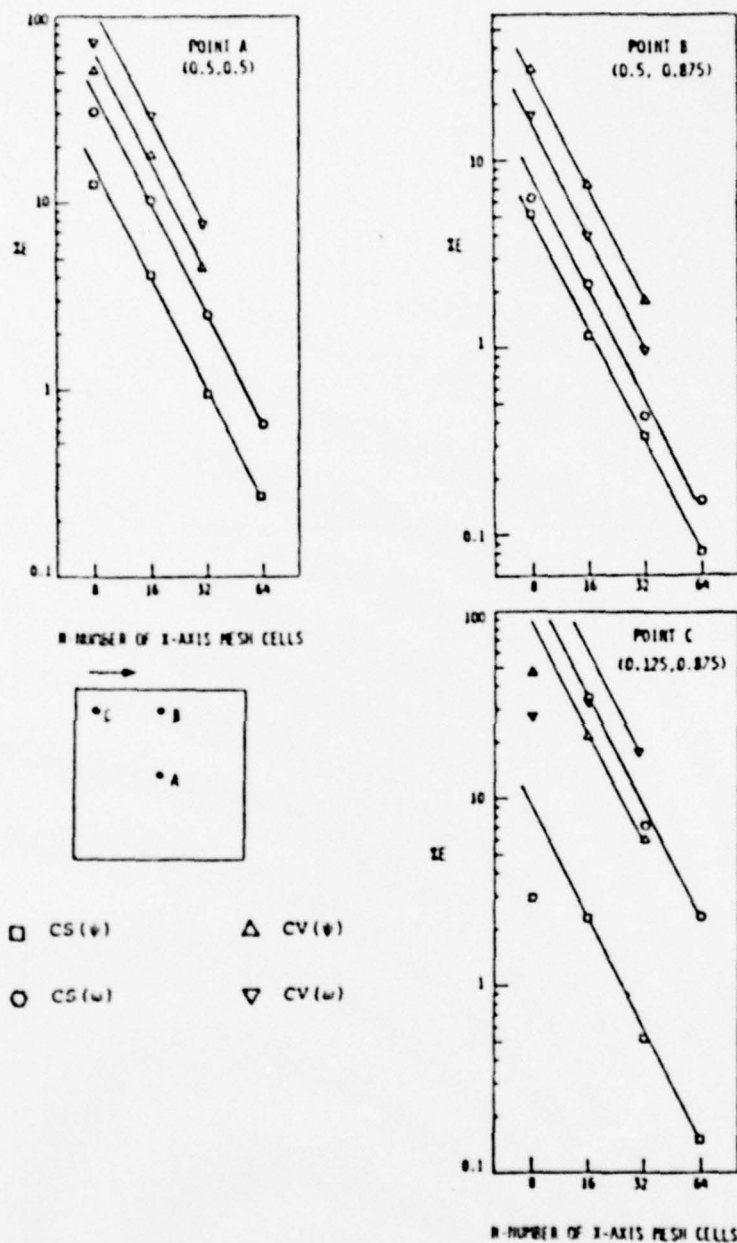FIG. 21. HORIZONTAL VELOCITY PROFILE THROUGH CAVITY CENTER.

FIG. 22. ACCURACY STUDY FOR (ω, ψ)-SYSTEM SOLUTIONS FOR DRIVEN-CAVITY FLOW.